Inventor
and Its Applications

David P. Madsen

Includes Student CD



Release 14

Inventor
and Its Applications

by

David P. Madsen
Instructor, Program Coordinator Drafting Technology
Northwest College, Powell, Wyoming
Autodesk Developer Network Member
SolidWorks Research Associate
American Design Drafting Association Member



Publisher
The Goodheart-Willcox Company, Inc.
Tinley Park, Illinois
www.g-w.com

Copyright © 2007
by
The Goodheart-Willcox Company, Inc.

All rights reserved. No part of this work may be reproduced, stored, or transmitted in any form or by any electronic or mechanical means, including information storage and retrieval systems, without the prior written permission of The Goodheart-Willcox Company, Inc.

Manufactured in the United States of America.

Library of Congress Catalog Card Number 2007011576

ISBN 978-1-59070-775-3

1 2 3 4 5 6 7 8 9 – 07 – 12 11 10 09 08 07

The Goodheart-Willcox Company, Inc. Brand Disclaimer: Brand names, company names, and illustrations for products and services included in this text are provided for educational purposes only and do not represent or imply endorsement or recommendation by the author or the publisher.

The Goodheart-Willcox Company, Inc. Safety Notice: The reader is expressly advised to carefully read, understand, and apply all safety precautions and warnings described in this book or that might also be indicated in undertaking the activities and exercises described herein to minimize risk of personal injury or injury to others. Common sense and good judgment should also be exercised and applied to help avoid all potential hazards. The reader should always refer to the appropriate manufacturer's technical information, directions, and recommendations; then proceed with care to follow specific equipment operating instructions. The reader should understand these notices and cautions are not exhaustive.

The publisher makes no warranty or representation whatsoever, either expressed or implied, including but not limited to equipment, procedures, and applications described or referred to herein, their quality, performance, merchantability, or fitness for a particular purpose. The publisher assumes no responsibility for any changes, errors, or omissions in this book. The publisher specifically disclaims any liability whatsoever, including any direct, indirect, incidental, consequential, special, or exemplary damages resulting, in whole or in part, from the reader's use or reliance upon the information, instructions, procedures, warnings, cautions, applications, or other matter contained in this book. The publisher assumes no responsibility for the activities of the reader.

Library of Congress Cataloging-in-Publication Data

Madsen, David P.
Inventor and its applications / by David P. Madsen.
p. cm.
Includes index.
ISBN 978-1-59070-775-3
1. Engineering graphics. 2. Engineering models–Data processing. 3. Autodesk Inventor (Electronic resource) I. Title.

T353.M153 2007
620.0042'0285536–dc22 2007011576



Introduction

Inventor and Its Applications is a text providing complete instruction in mastering fundamental Autodesk Inventor® tools and modeling techniques. Typical applications of Inventor are presented with basic modeling and design concepts. The topics are covered in an easy-to-understand sequence and progress in a way that allows you to become comfortable with the tools as your knowledge builds from one chapter to the next. In addition, ***Inventor and Its Applications*** offers the following features:

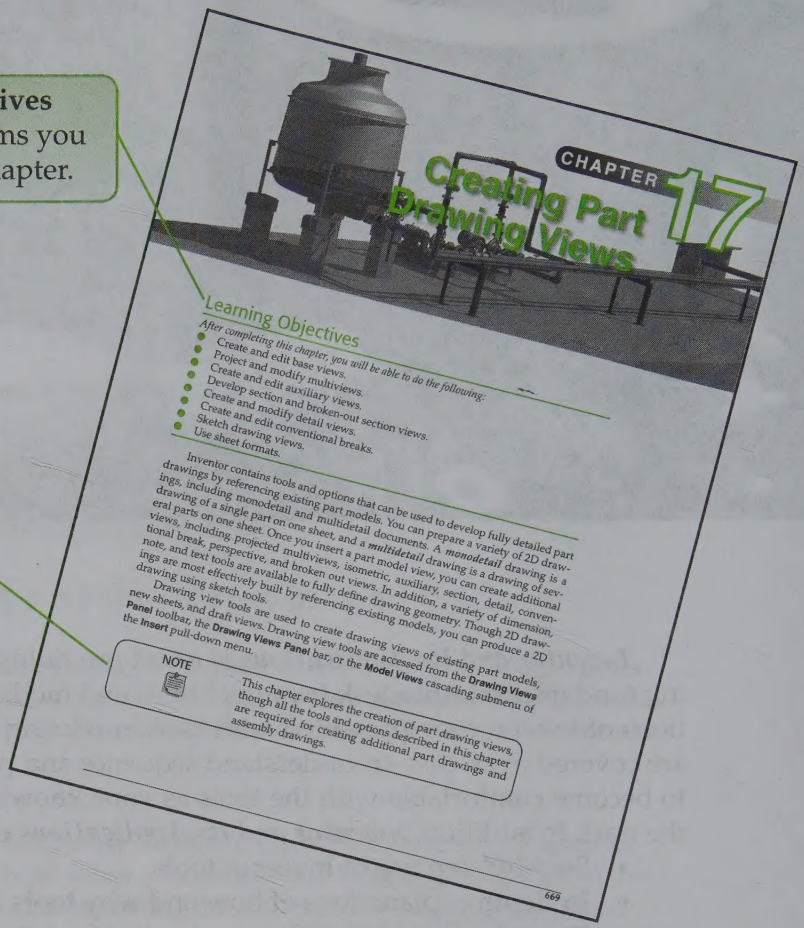
- Step-by-step use of Inventor tools.
- In-depth explanations of how and why tools function as they do.
- Extensive use of font changes to specify certain meanings.
- Examples and discussions of industry practices and standards.
- Actual screen captures of Inventor and Windows features and functions.
- Professional Tips explaining how to use Inventor effectively and efficiently.
- Over two hundred exercises to reinforce the chapter topics. These exercises also build on previously learned material.
- Chapter tests for review of tools and key Inventor concepts.
- A large selection of modeling and drafting problems supplementing each chapter. Problems are presented as 3D illustrations, industrial drawings, and engineering sketches.

Learning Objectives identify the key items you will learn in the chapter.

Notes explain important aspects of a topic.

Professional Tips increase your productivity in using Inventor tools and techniques.

Exercise References identify when you should complete an Exercise from the Student CD.



CHAPTER 17

Creating Part Drawing Views

Learning Objectives

After completing this chapter, you will be able to do the following:

- Create and edit base views.
- Project and modify multiviews.
- Create and edit auxiliary views.
- Develop section and broken-out section views.
- Create and modify detail views.
- Create and edit conventional breaks.
- Sketch drawing views.
- Use sheet formats.

Inventor contains tools and options that can be used to develop fully detailed part drawings by referencing existing part models. You can prepare a variety of 2D drawings, including monodetail and multidetail documents. A ***monodetail*** drawing is a drawing of a single part on one sheet, and a ***multidetail*** drawing is a drawing of several parts on one sheet. Once you insert a part model view, you can create additional views, including projected multiviews, isometric, auxiliary, section, detail, conventional break, perspective, and broken out views. In addition, a variety of dimension, note, and text tools are available to fully define drawing geometry. Though 2D drawings are most effectively built by referencing existing models, you can produce a 2D drawing using sketch tools.

Drawing view tools are used to create drawing views of existing part models, new sheets, and draft views. Drawing view tools are accessed from the **Drawing Views Panel** bar, or the **Model Views** cascading submenu of the **Insert** pull-down menu.

NOTE

This chapter explores the creation of part drawing views. Though all the tools and options described in this chapter are required for creating additional part drawings and assembly drawings.

669

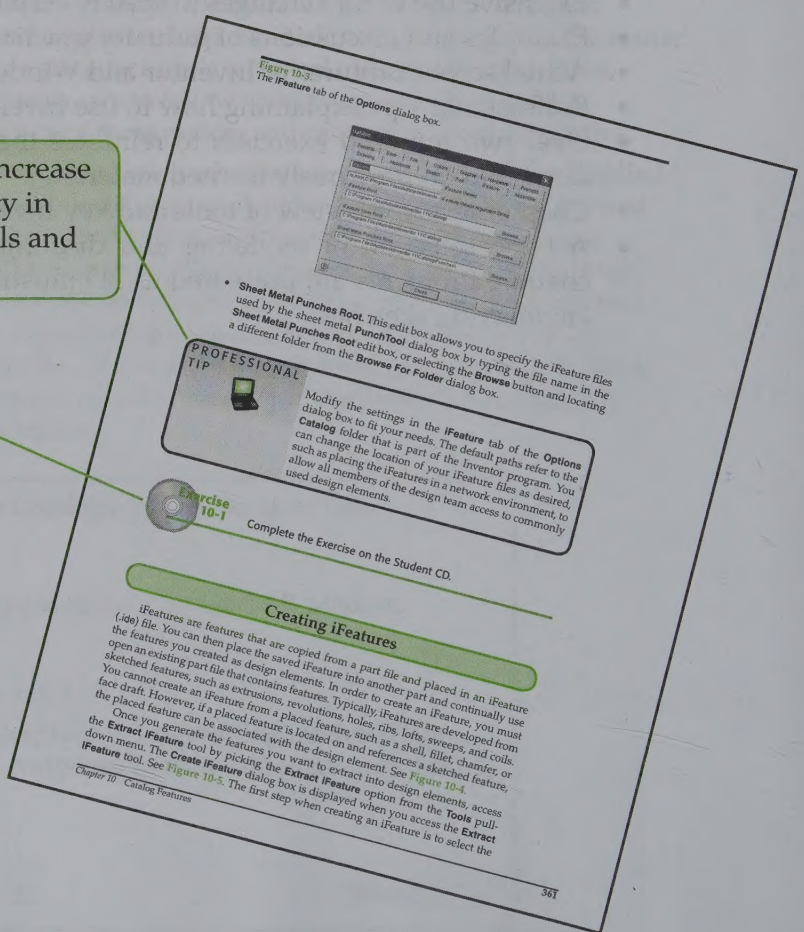


Figure 10-3
The **iFeature** tab of the **Options** dialog box.

- **Sheet Metal Punches Root.** This edit box allows you to specify the iFeature files used by the sheet metal **PunchTool** dialog box by typing the file name in the **Sheet Metal Punches Root** edit box, or selecting the **Browse** button and locating a different folder from the **Browse For Folder** dialog box.

PROFESSIONAL TIP

Modify the settings in the **iFeature** tab of the **Options** dialog box to fit your needs. The default paths refer to the **Catalog** folder that is part of the Inventor program. You can change the location of your iFeature files as desired, such as placing the iFeatures in a network environment, to allow all members of the design team access to commonly used design elements.

Exercise 10-1

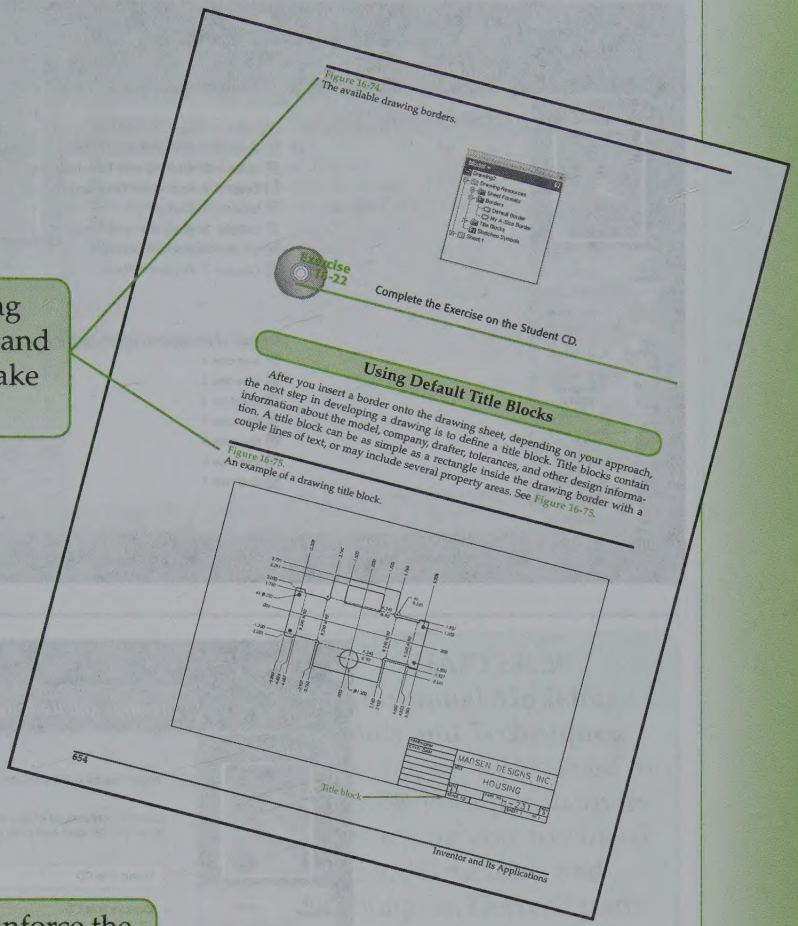
Complete the Exercise on the Student CD.

Creating iFeatures

iFeatures are features that are copied from a part file and placed in an iFeature (.ide) file. You can then place the saved iFeature into another part and continually use the features you created as design elements. In order to create an iFeature, you must open an existing part file that contains features. Typically, iFeatures are developed from sketched features, such as extrusions, revolutions, holes, ribs, lofts, sweeps, and coils. You cannot create an iFeature from a placed feature, such as a shell, fillet, chamfer, or face draft. However, if a placed feature is located on and references a sketched feature, the placed feature can be associated with the design element. See Figure 10-4.

Once you generate the features you want to extract into design elements, access the **Extract iFeature** tool by picking the **Extract iFeature** option from the **Tools** pull-down menu. The **Create iFeature** dialog box is displayed when you access the **Extract iFeature** tool. See Figure 10-5. The first step when creating an iFeature is to select the

Illustrations, including Inventor "screen shots" and line art illustrations, make learning easy.


Figure 16-74.
The available drawing borders.

Exercise 16-22
Complete the Exercise on the Student CD.

Using Default Title Blocks

After you insert a border onto the drawing sheet, depending on your approach, the next step in developing a drawing is to define a title block. Title blocks contain information about the model, company, drafter, tolerances, and other design information. A title block can be as simple as a rectangle inside the drawing border with a couple lines of text, or may include several property areas. See Figure 16-75.

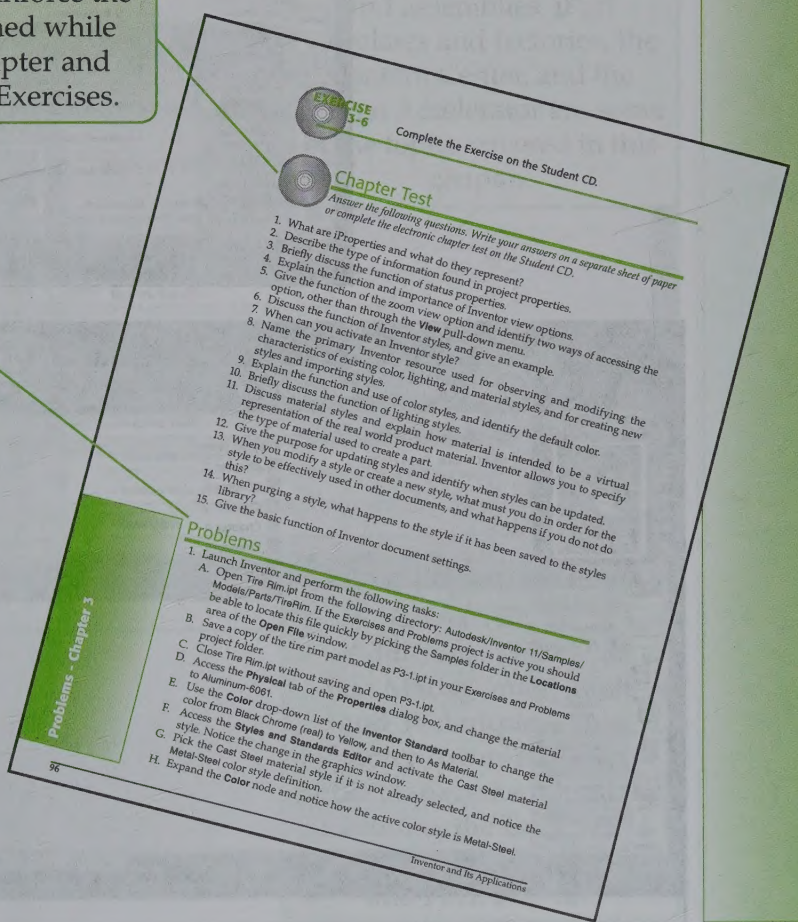
Figure 16-75.
An example of a drawing title block.

654

Inventor and Its Applications

Chapter Tests reinforce the knowledge gained while reading the chapter and completing the Exercises.

Drawing Problems require application of chapter concepts and problem-solving techniques. Icons identify problems from various drafting disciplines.


Exercise 3-6
Complete the Exercise on the Student CD.

Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. What are iProperties and what do they represent?
2. Describe the type of information found in project properties.
3. Briefly discuss the function of status properties.
4. Explain the function and importance of Inventor view options.
5. Give the function of the zoom view option and identify two ways of accessing the option, other than through the **View** pull-down menu.
6. Discuss the function of Inventor styles, and give an example.
7. When can you activate an Inventor style?
8. Name the primary Inventor resource used for observing and modifying the characteristics of existing color, lighting, and material styles, and for creating new styles and importing styles.
9. Explain the function and use of color styles, and identify the default color.
10. Briefly discuss the function of lighting styles.
11. Discuss material styles and explain how material is intended to be a virtual representation of the real world product material. Inventor allows you to specify the type of material used to create a part.
12. Give the purpose for updating styles and identify when styles can be updated.
13. When you modify a style or create a new style, what must you do in order for the style to be effectively used in other documents, and what happens if you do not do this?
14. When purging a style, what happens to the style if it has been saved to the styles library?
15. Give the basic function of Inventor document settings.

Problems

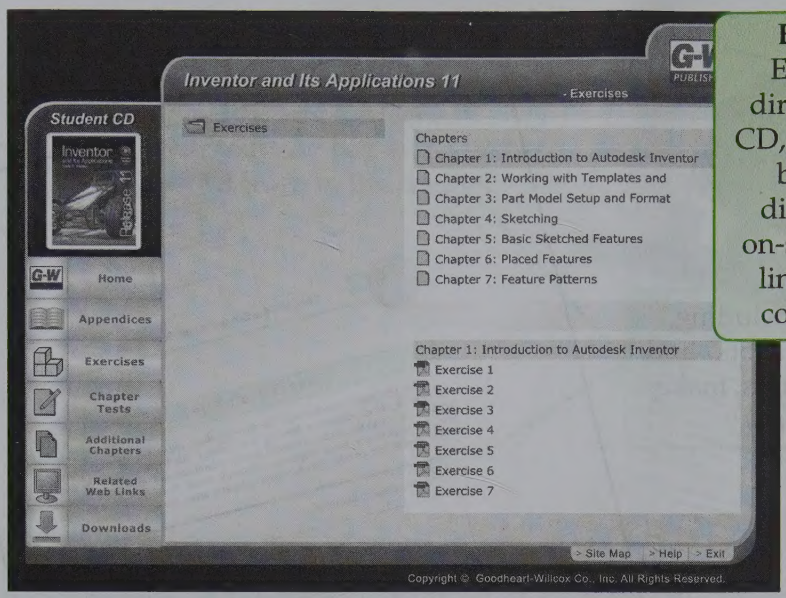
1. Launch Inventor and perform the following tasks:
   A. Open *Tire Rim.ipt* from the following directory: Autodesk/Inventor 11/Samples/Models/Parts/TireRim. If the Exercises and Problems project is active you should be able to locate this file quickly by picking the Samples folder in the **Locations** area of the **Open File** window.
   B. Save a copy of the tire rim part model as P3-1.ipt in your Exercises and Problems project folder.
   C. Close Tire Rim.ipt without saving and open P3-1.ipt.
   D. Access the **Physical** tab of the **Properties** dialog box, and change the material to Aluminum-6061.
   E. Use the **Color** drop-down list of the **Inventor Standard** toolbar to change the color from Black Chrome (real) to Yellow, and then to As Material.
   F. Access the **Styles and Standards Editor** and activate the Cast Steel material style. Notice the change in the graphics window.
   G. Pick the Cast Steel material style if it is not already selected, and notice the Metal-Steel color style definition.
   H. Expand the **Color** node and notice how the active color style is Metal-Steel.

Problems - Chapter 3

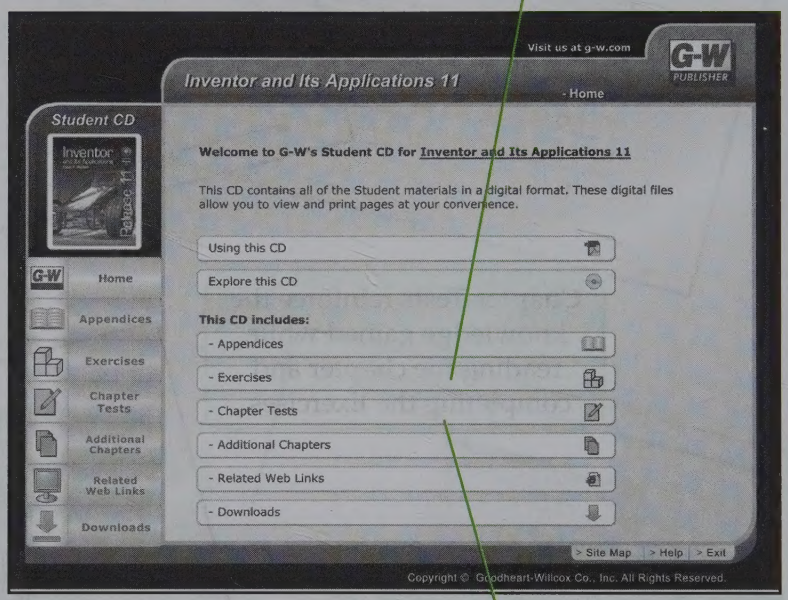
96

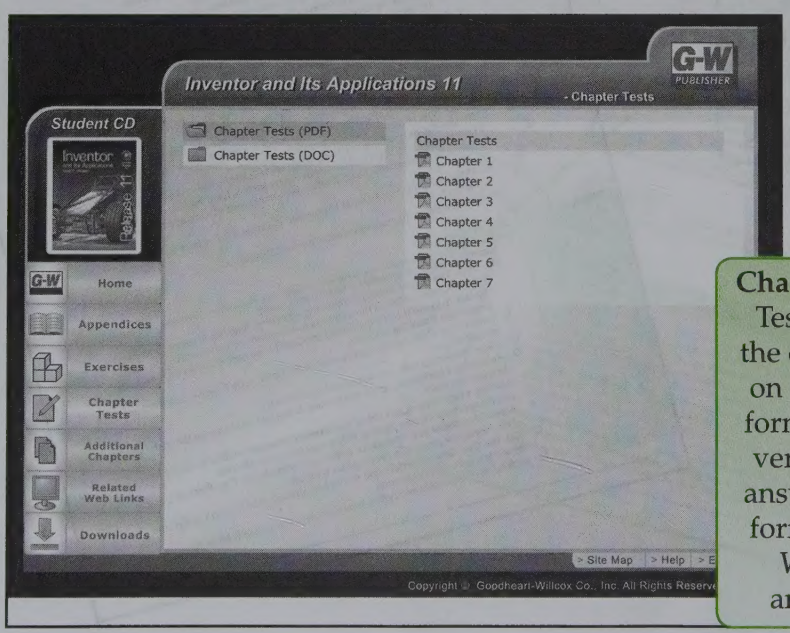
Inventor and Its Applications

Features of the Textbook

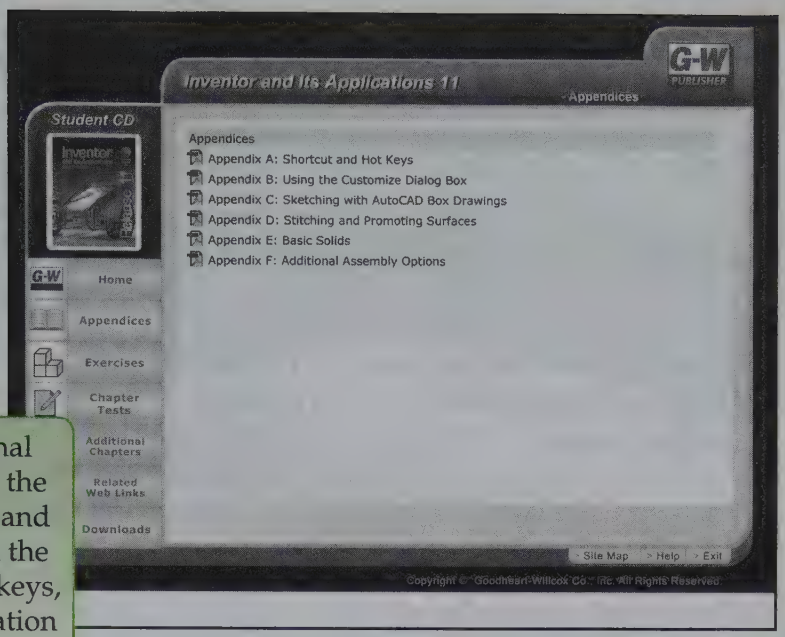


Exercises. In-chapter Exercises are accessed directly from the Student CD, allowing you to switch between the Exercise directions and Inventor on-screen. Many Exercises link to Inventor files for completing the activity.



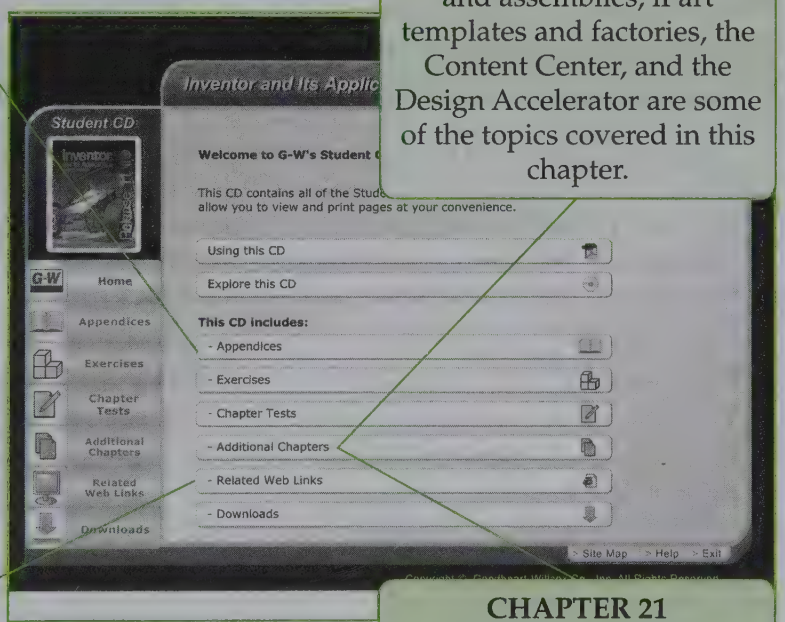


Chapter Tests. The Chapter Tests found at the end of the chapters are also found on the Student CD in two formats. Print out the PDF version and write in the answers, or open the DOC form version in Microsoft Word and enter your answers electronically.



Appendices. Additional material is provided in the appendices to enhance and reinforce the content in the text. There are shortcut keys, hot keys, and customization methods to increase efficiency; procedures for working with AutoCAD drawings and basic solids; and techniques to stitch and promote surfaces.

CHAPTER 20 Additional Modeling Tools and Techniques. This chapter is designed to increase your productivity by teaching you advanced modeling tools and techniques. Derived parts and assemblies, iPart templates and factories, the Content Center, and the Design Accelerator are some of the topics covered in this chapter.



Related Web Links. Use this to access a wide variety of CAD/drafting Web sites.

CHAPTER 21 Additional Data and File Management Tools and Techniques. This chapter covers advanced file management topics including the Engineer's Notebook, Design Assistant, and Pack and Go.

Fonts Used in This Text

Different typefaces are used throughout this text to define terms and identify Inventor tools. The following typeface conventions are used in this textbook:

Text Element	Example
Inventor menus, tools, dialog box names, and button names	When you select **Spline Tension**, the **Spline Tension** dialog box is displayed.
File names, folder names, paths, and keyboard-entry items	The first part, SHOCK_BODY.ipt, shown in Figure 12-37, contains four iMates.
Important terms	A ***decal*** is any type of image applied to a part or assembly and is used to display information or decorate a product.
Keyboard keys	Press [Enter] to continue.

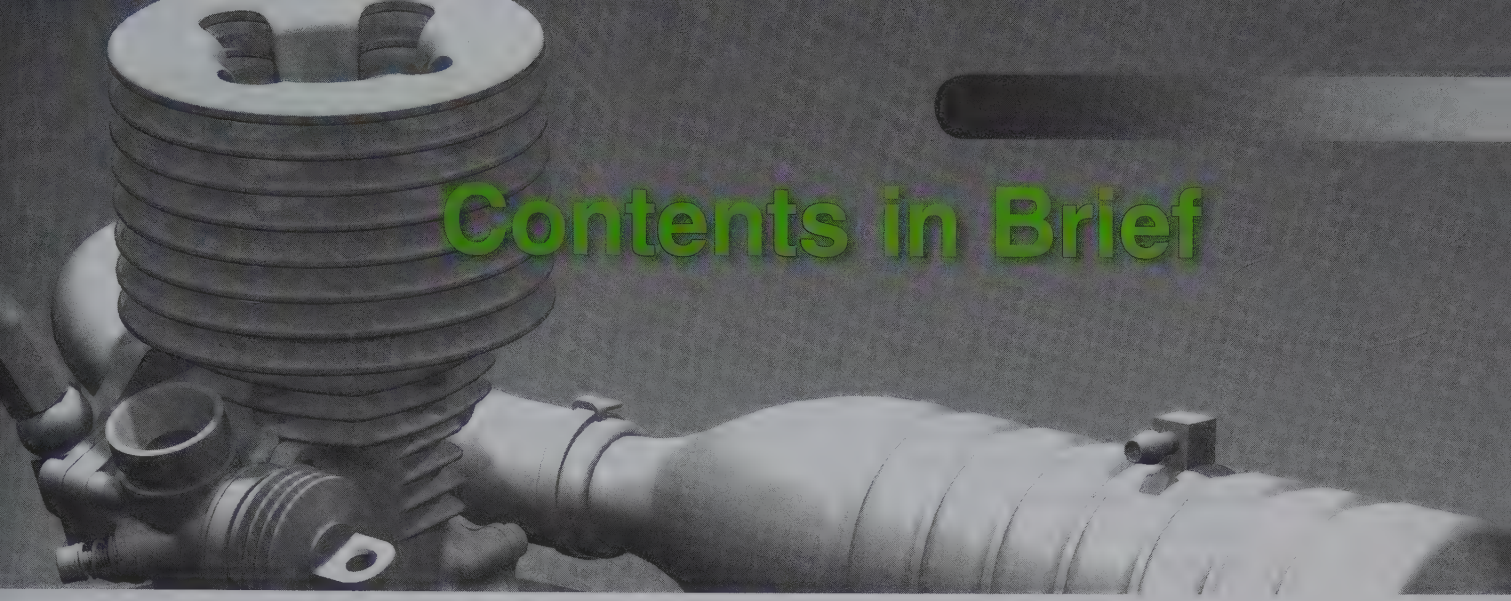
Other Text References

For additional information, standards from organizations, such as ANSI (American National Standards Institute) and ASME (American Society of Mechanical Engineers), are referenced throughout the text. These standards are used to help you create drawings that follow industrial, national, and international practices. The Student CD includes a list of many of these standards.

Trademarks

Autodesk, AutoCAD, Inventor, DWG, DXF, and the Autodesk logo are registered trademarks or trademarks of Autodesk, Inc., in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders.

Microsoft, Windows, and Windows XP are registered trademarks of Microsoft Corporation in the United States and/or other countries.



Contents in Brief

1	Introduction to Autodesk Inventor	17
2	Working with Templates and Projects	53
3	Part Model Setup and Format	67
4	Sketching	99
5	Basic Sketched Features	157
6	Placed Features	209
7	Feature Patterns	253
8	Work Features	279
9	Complex Features	307
10	Catalog Features	359
11	Sheet Metal Parts	381
12	Assemblies	431
13	Weldments	509
14	Adaptive Models	539
15	Presentations	557
16	Drawing Setup and Format	579
17	Creating Part Drawing Views	669
18	Dimensioning Part Drawings	721
19	Assembly, Weldment, and Multiple Sheet Drawings	779
	Index	823

About the Author

David P. Madsen is a Computer Aided Design and Drafting consultant and educator providing technical services and training for private, education, and industry clients. Dave holds a Master of Science degree in Educational Policy, Foundations, and Administrative Studies with a specialization in Postsecondary, Adult, and Continuing Education; a Bachelor of Science degree in Technology Education; and an Associate of Science degree in General Studies and Drafting Technology.

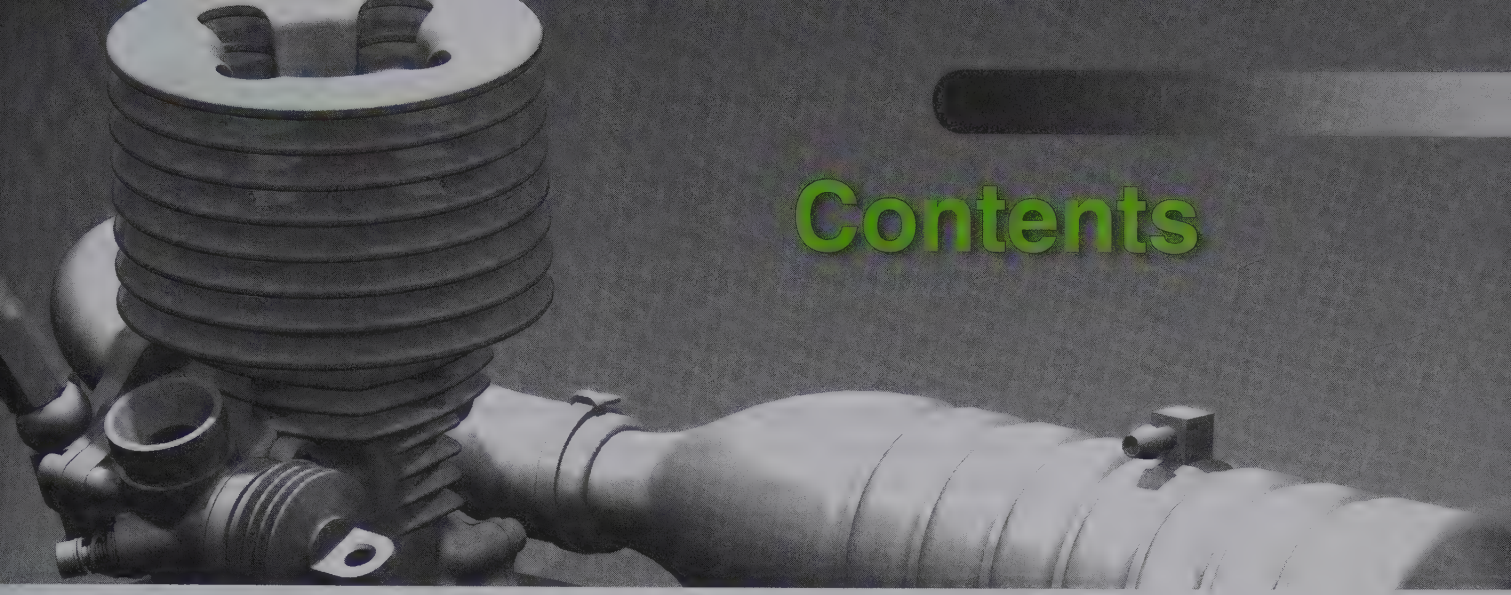
Dave has been involved in providing drafting and computer aided design and drafting (CADD) instruction to adult learners since 1999. He is a former college Drafting Technology Instructor and Program Coordinator. Dave has extensive and varied experience in the drafting, design, and engineering fields. He has worked in the drafting industry for over ten years and has created everything from mechanical and electronic to architectural and civil drawings.

Acknowledgments

I would like to give special thanks to my father, David A. Madsen, for his contributions, encouragement, and professional support during the development of this textbook.

About the Cover

Bill Bogan comes from a family of race car drivers and enthusiasts. Bill used Inventor to create this model of a three quarter midget race car based on a race car his dad, Robert (Bob) Bogan, built. Read more about this model and the Bogan racers by picking the Bogan Racing link at www.b2-design.biz/links.html.



Contents

CHAPTER 1

Introduction to Autodesk Inventor 17

Inventor Files 17
Parametric Solid Modeling 21
Part Model Elements 22
Constraints 25
The Inventor Interface 26
Starting a New File 37
Controlling Windows 39
Saving Your Work 42
Opening Existing Files 44
Getting Help 47
Chapter Test 49
Problems 50

CHAPTER 2

Working with Templates and Projects 53

Templates 53
Projects 56
Chapter Test 66
Problems 66

CHAPTER 3

Part Model Setup and Format 67

iProperties 67
View Options 72
Styles and Standards 77
Document Settings 89
Application Options 89
Chapter Test 96
Problems 96

CHAPTER 4
Sketching 99
Sketch Concepts 99
Beginning a Sketch 101
Finishing a Sketch 101
Referencing and Projecting Geometry 102
Sketching Lines 103
Sketching Splines 106
Sketching Circles and Ellipses 109
Sketching Arcs 111
Sketching Rectangles Using Rectangle Tools 114
Adding Sketch Fillets, Rounds, and Chamfers 115
Adding Sketch Points and Hole Centers 118
Sketching Regular Polygons Using the Polygon Tool 119
Adding Sketch Text 120
Patterning Sketch Geometry 122
Offsetting Sketch Geometry 127
Constraining Sketches Using Geometric Constraints 128
Constraining Sketches Using Dimensions 131
Sketching with the Precise Input Toolbar 140
Sketch Parameters 141
Using Sketch Editing Tools 143
Using the Sketch Doctor 147
Chapter Test 149
Problems 151

CHAPTER 5
Basic Sketched Features 157
Creating Extruded Base Features 157
Creating Revolved Base Features 162
Developing a Coil 164
Sketching on a Base Feature 171
Creating Additional Extrusions 173
Creating Additional Revolutions and Coils 179
Using Open Sketch Profiles 180
Creating Holes 182
Creating Ribs and Webs 191
Chapter Test 195
Problems 196

CHAPTER 6
Placed Features 209
Adding Fillets and Rounds 210
Adding Chamfers 222
Shelling Features 226
Applying Face Draft 230
Placing Threads 233
Thickening and Offsetting 236
Moving Faces 239
Chapter Test 242
Problems 243

CHAPTER 7 Feature Patterns 253
Mirroring Features 254
Creating Rectangular Feature Patterns 257
Creating Circular Feature Patterns 263
Chapter Test 268
Problems 269

CHAPTER 8 Work Features 279
Using Work Planes 279
Projecting Cut Edges 290
Using Work Axes 290
Using Work Points 294
Chapter Test 300
Problems 300

CHAPTER 9 Complex Features 307
Creating Lofts 307
Creating Basic Sweeps 317
Creating Advanced Lofts and Sweeps 323
Using 3D Sketch Tools 324
Creating Split Features 332
Embossing and Engraving 335
Additional Surface Tools and Techniques 339
Creating Decals 341
Chapter Test 346
Problems 347

CHAPTER 10 Catalog Features 359
Viewing and Managing iFeatures 360
Creating iFeatures 361
Inserting iFeatures 366
Editing iFeatures 368
Creating Table-Driven iFeatures 369
Inserting Table-Driven iFeatures 376
Chapter Test 377
Problems 378

CHAPTER 11 Sheet Metal Parts 381
Sheet Metal Setup and Format 381
Sheet Metal Styles 381
Sheet Metal Sketched Features 388
Sheet Metal Placed Features 404
Using the Punch Tool 421
Creating Flat Patterns 424
Chapter Test 426
Problems 427

CHAPTER 12

Assemblies 431

Developing Assembly Models 432
Creating Assemblies Using Existing Components 433
Developing Components In-Place 437
Constraining Assemblies 442
Patterning Components 472
Mirroring Components 477
Copying Components 480
Adjusting Components 483
Creating Flexible Subassemblies 491
Using Design View Representations 492
Using Positional Representations 494
Managing Large Assemblies 498
Chapter Test 500
Problems 501

CHAPTER 13

Weldments 509

Developing Weldment Models 510
Material Preparation 511
Welding 514
Machining Weldments 529
Converting an Assembly to a Weldment 530
Editing Weldments 532
Creating Weld Bead Reports 533
Chapter Test 535
Problems 536

CHAPTER 14

Adaptive Models 539

Creating Adaptive Sketches 540
Developing Adaptive Features 544
Creating Adaptive Assembly Parts 548
Developing Adaptive Subassemblies 551
Chapter Test 553
Problems 554

CHAPTER 15

Presentations 557

Developing Presentation Models 558
Creating Presentation Views 559
Exploding Presentations 561
Using the Precise View Rotation Tool 569
Animating Presentations 570
Placing Additional Presentation Views 575
Chapter Test 576
Problems 577

CHAPTER 16
Drawing Setup and Format 579
Developing Drawings 580
Styles and Standards 581
Document Settings 683
Application Options 643
Drawing Sheet Parameters 645
Using Default Borders 646
Creating a Custom Border 649
Using Default Title Blocks 654
Creating a Custom Title Block 657
Chapter Test 658
Problems 659

CHAPTER 17
Creating Part Drawing Views 669
Creating Base Views 670
Projecting Views 679
Creating Auxiliary Views 682
Developing Section Views 685
Developing Broken-out Views 693
Creating Detail Views 699
Creating Conventional Breaks 703
Sketching Views 708
Using Sheet Formats 709
Chapter Test 710
Problems 711

CHAPTER 18
Dimensioning Part Drawings 721
Working with Model Dimensions 722
Drawing Dimensions 724
Chapter Test 773
Problems 774

CHAPTER 19
Assembly, Weldment, and Multiple Sheet Drawings 779
Assembly Drawings 779
Weldment Drawings 789
Dimensioning Assembly and Weldment Drawings 790
Multiple Sheet Drawings 814
Printing Drawings 817
Chapter Test 820
Problems 821

Index 823

Student CD Contents

Using the Student CD

Exercises

Chapter Tests

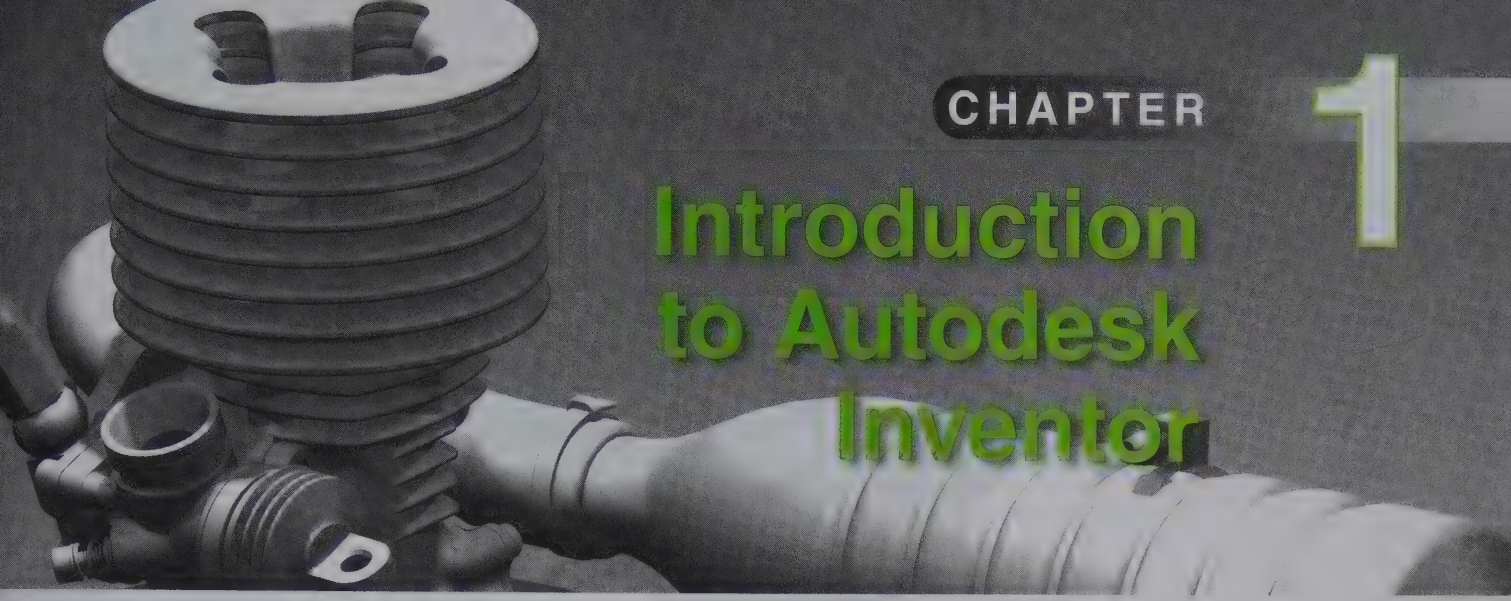
Appendices

Appendix A— Shortcut and Hot Keys
Appendix B— Using the Customize Dialog Box
Appendix C— Sketching with AutoCAD Drawings
Appendix D—Stitching and Promoting Surfaces
Appendix E— Basic Solids
Appendix F— Additional Assembly Options

Additional Chapters

Chapter 20— Additional Modeling Tools and Techniques
Chapter 21— Additional Data and File Management Tools and Techniques

Related Web sites



CHAPTER 1

Introduction to Autodesk Inventor

Learning Objectives

After completing this chapter, you will be able to do the following:

- Describe the four primary Autodesk Inventor file types.
- Explain the concept of parametric modeling.
- Define the elements of a part model.
- Describe the basic steps involved in developing parametric models.
- Explain the purpose of constraints and parametric associations.
- Describe the Autodesk Inventor user interface.
- Recognize and apply interface items.
- Start new files and control file windows.
- Save your work and open existing files.
- Locate help files and other help resources.

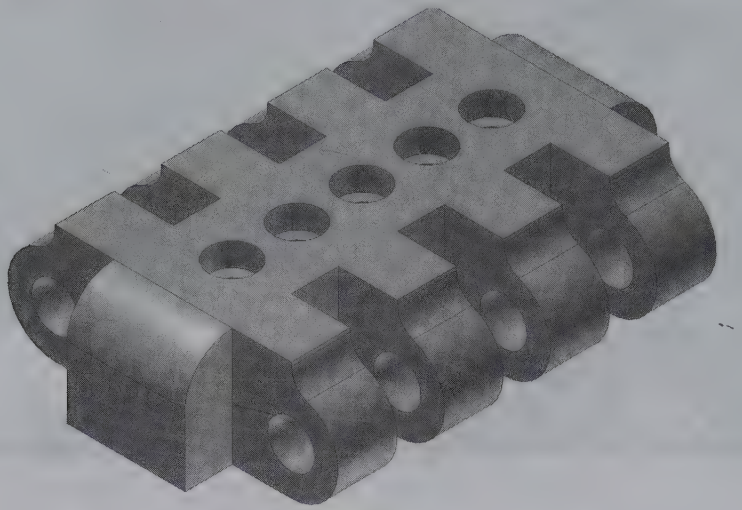
Autodesk Inventor combines three-dimensional (3D) solid modeling with two-dimensional (2D) drawing capabilities. See Figure 1-1. Autodesk Inventor, referred to as *Inventor* for the rest of this book, is a very powerful computer-aided design and drafting (CADD) system that allows you to start with very basic product design ideas and end with a virtual prototype and complete set of working drawings.

Inventor Files

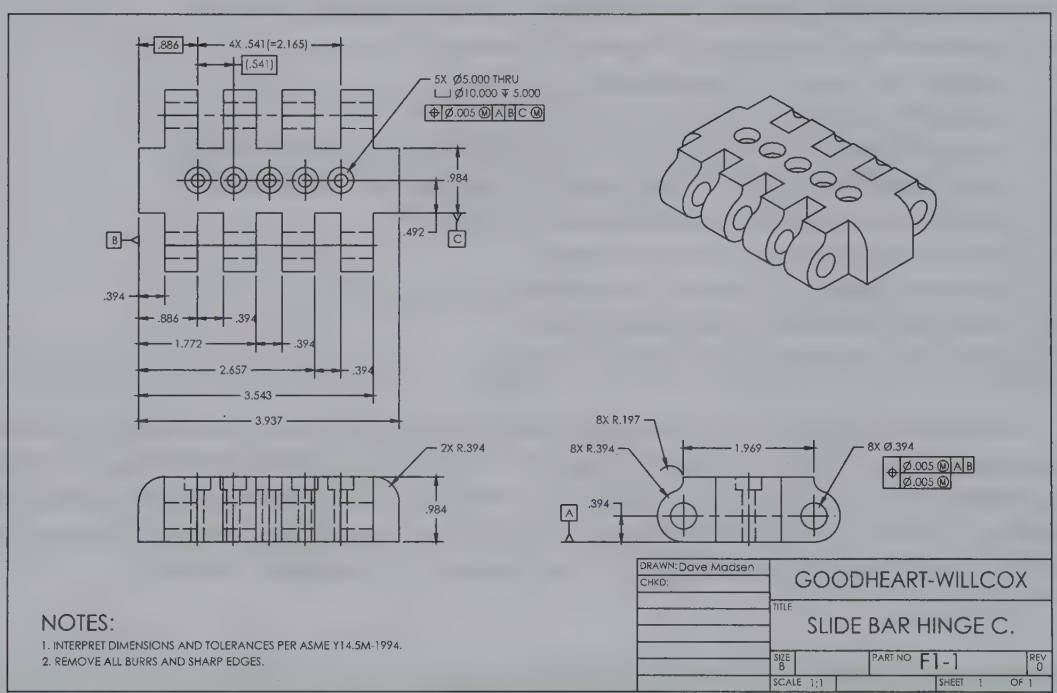
Inventor functions using four primary file types: part, assembly, presentation, and drawing. Each file serves a specific design and documentation purpose depending on the application. In some cases you may only work with a single part file or a part and drawing file, while other projects can require that you produce hundreds of parts, assemblies, presentations, and drawings. The following paragraphs briefly describe each type of Inventor file:

- **Part files** carry the extension .ipt, which stands for *Inventor part*. For example, a part named Part 1 is Part1.ipt. Part files are used to create single parts, sheet metal parts, component parts for an assembly or weldment, and geometry to be used in two-dimensional drawings. A part can be created separately in an individual part file template or can be created in an assembly or weldment file template with other components. An example of a part is shown in Figure 1-2.

Figure 1-1.
Inventor has tools to create three-dimensional solid models and two-dimensional drawings.



3D model



2D drawing

- **Assembly files** are used to create assemblies and subassemblies, and reference multiple part and assembly files. An ***assembly*** is a grouping of components. The term ***component*** is used to describe the individual parts and subassemblies used to create an assembly. A ***subassembly*** is an assembly that is added to another assembly. An assembly can be built by inserting existing components into an assembly file or by creating individual components in an assembly file. Assembly files have the extension .iam, which stands for *Inventor assembly*. For example, an assembly named Assembly1 is Assembly1.iam. The C-clamp shown in Figure 1-3 is an example of an assembly file.

Figure 1-2.
This is an example of a part file.

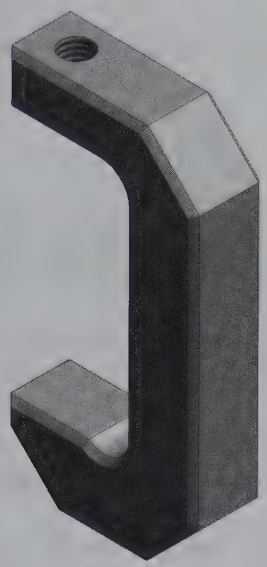
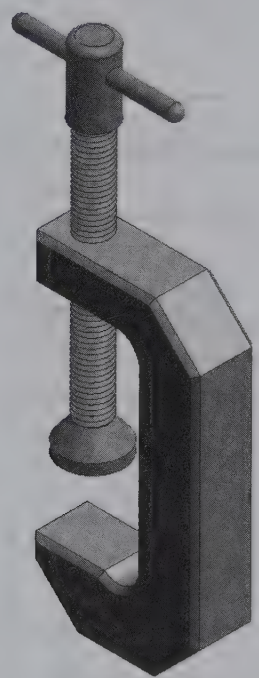


Figure 1-3.
An assembly file, such as this, is a collection of part files.



- **Presentations** allow you to display assemblies in a way that shows how separate parts and subassemblies interact together within the full assembly. Presentation files are used to create exploded, animated, and stylized assembly views. Presentations may include connection graphics between parts, known as ***trails***, and part modification, or ***tweaking***, of components. Presentation files have an .ipn extension, which stands for *Inventor presentation*. For example, a presentation named Presentation1 is Presentation1.ipn. The exploded view of the C-clamp shown in Figure 1-4 is an example of a presentation file.
- **Drawings** are representations of models. Drawing files have the extension .idw, which stands for *Inventor drawing*. For example, a drawing named Drawing1 is Drawing1.idw. An example of a drawing for a C-clamp body is shown in Figure 1-5.

Figure 1-4.
This presentation file contains trails to show how the assembly is built.

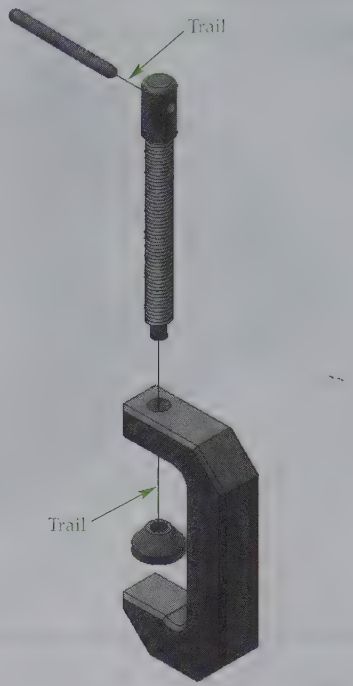
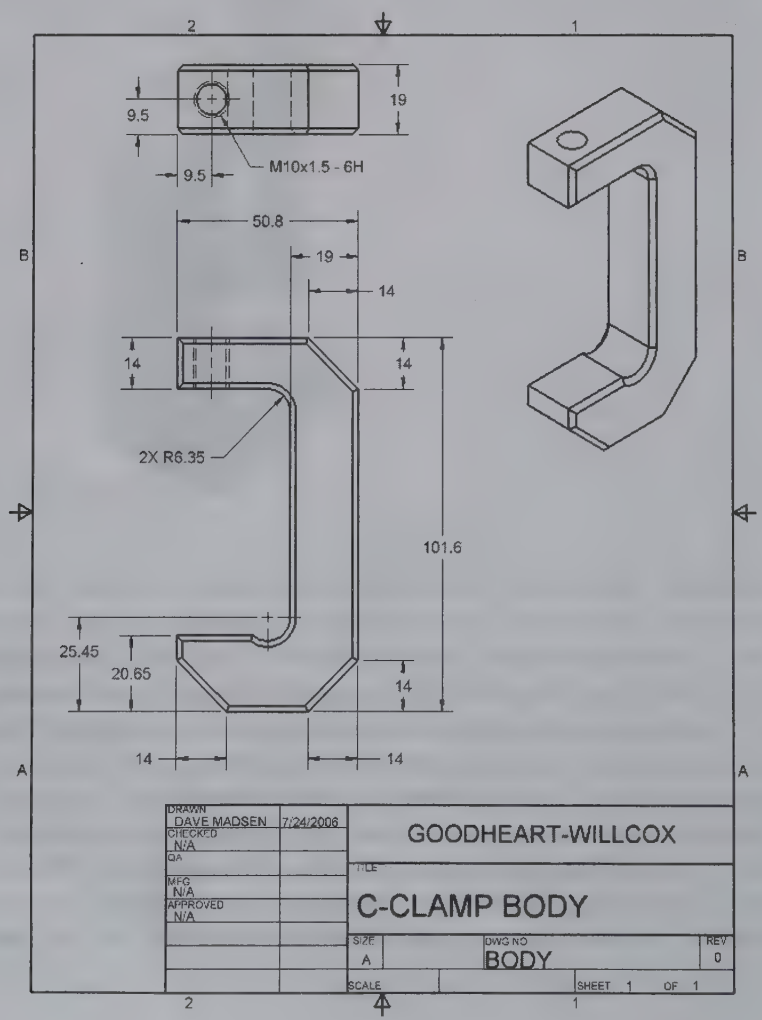


Figure 1-5.
A drawing file should contain enough information to completely describe the part or assembly.





Complete the Exercise on the Student CD.

Parametric Solid Modeling

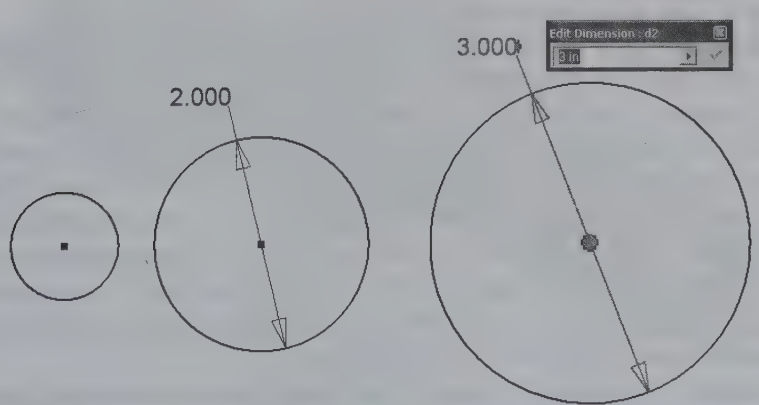
Inventor works on the concept of ***parametric solid modeling***. *Parametric* means that models contain parameters that control the size, shape, and position of model geometry. Parameters can be adjusted to make changes to the model, but if the modifications conflict with any other geometry, the model cannot be built without errors. *Solid models* contain object volume and mass data that can be used to analyze internal and external object characteristics.

The parametric concept is also referred to as ***intelligence***, because parametric modeling occurs as a result of Inventor's ability to store and manage model information. Everything associated with a model is stored in a database. The database contains knowledge of every model characteristic, including calculations, sketches, features, dimensions, geometric constraints, when each piece of the model was created, and all other model parameters and properties.

The easiest way to understand the basic principle of parametric association is to study the size of a parametrically created geometric shape. For example, the first circle shown in **Figure 1-6** was drawn with a diameter of 1". When a dimension is added to the circle that defines the circle's diameter as 2", the size of the circle automatically changes according to the 2" diameter dimension. When a new diameter of 3" is applied to the circle, the size of the circle automatically changes from two to three inches in diameter. This example shows how the size of the circle is associated with the circle dimension. The circle only increases in size when the parametric value of the dimension is edited. Also notice how the circle centerlines automatically expand in length when the circle gets larger. This is another example of a basic parametric association.

Parametric modeling allows you to control every aspect of a model and drawing during and after the design and documentation process. In the parametric modeling world of Inventor, you begin a design with a simple 2D sketched shape and develop it into a complex 3D model. It is important to think about the parametric relationships

Figure 1-6.
After drawing the circle, you can control its size by attaching a dimension.



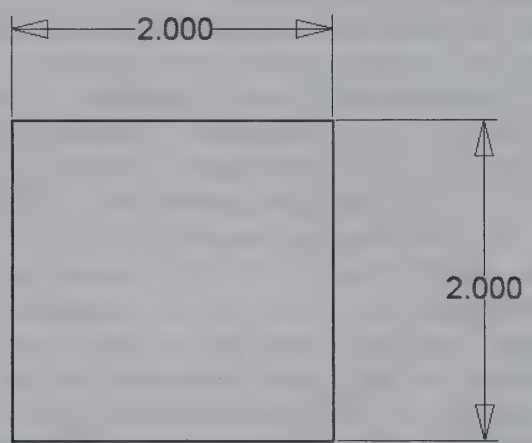
between model and drawing elements and the entire design process. Planning how to build a model and thinking about how it relates to future parts, assemblies, and drawings is an important step to beginning a design.

Part Model Elements

Part models serve as the primary solid model, and are used to form subassemblies and assemblies. Typically every part model contains at least one sketch and at least one sketched feature, while a more complex model can include placed, work, pattern, and catalog features. Sketching is the first step in the creation of a model. ***Sketches***, like the sketch shown in Figure 1-7, provide the foundation, guide, and profile for developing sketched features. ***Sketched features*** include extrusions, revolutions, sweeps, lofts, and coils, depending on the design and sketch geometry used to create the feature. Most of the time, the initial feature, known as the ***base feature***, is produced as a sketched feature. Ribs, embossments, and some holes are also sketched features. See Figure 1-8.

Features added to an existing feature without using a sketch are called ***placed features***. Shells, fillets, chamfers, threads, and face drafts are considered placed features. See Figure 1-9. Generating placed features requires adding size dimensions

Figure 1-7.
This simple sketch has two dimensions that control the object's size.



and selecting a location, such as a point or an edge. Often placed features are created toward the end of the design process.

Work features are used to direct the location and arrangement of other features. Work features are used for construction and reference purposes in order to develop additional sketches and features in areas where no geometry is available or can be created. As shown in Figure 1-10, Inventor work features include work planes, work axes, and work points. ***Work planes*** are used to create construction planes. ***Work axes*** are used to create construction lines and axes. ***Work points*** are used to create construction points. Work features may be placed anywhere on a feature or in three-dimensional space, and can be parametrically associated with a model and change when a feature or sketch is modified. In the same respect, if a work feature is modified, features attached to the work feature also reflect the changes.

Catalog features are similar to placed features, because you can place a catalog feature onto an existing feature. However, catalog features are often far more complex than a standard shell, fillet, chamfer, or face draft. See Figure 1-11. Catalog features

Figure 1-8.
These sketched features are added to the base feature.

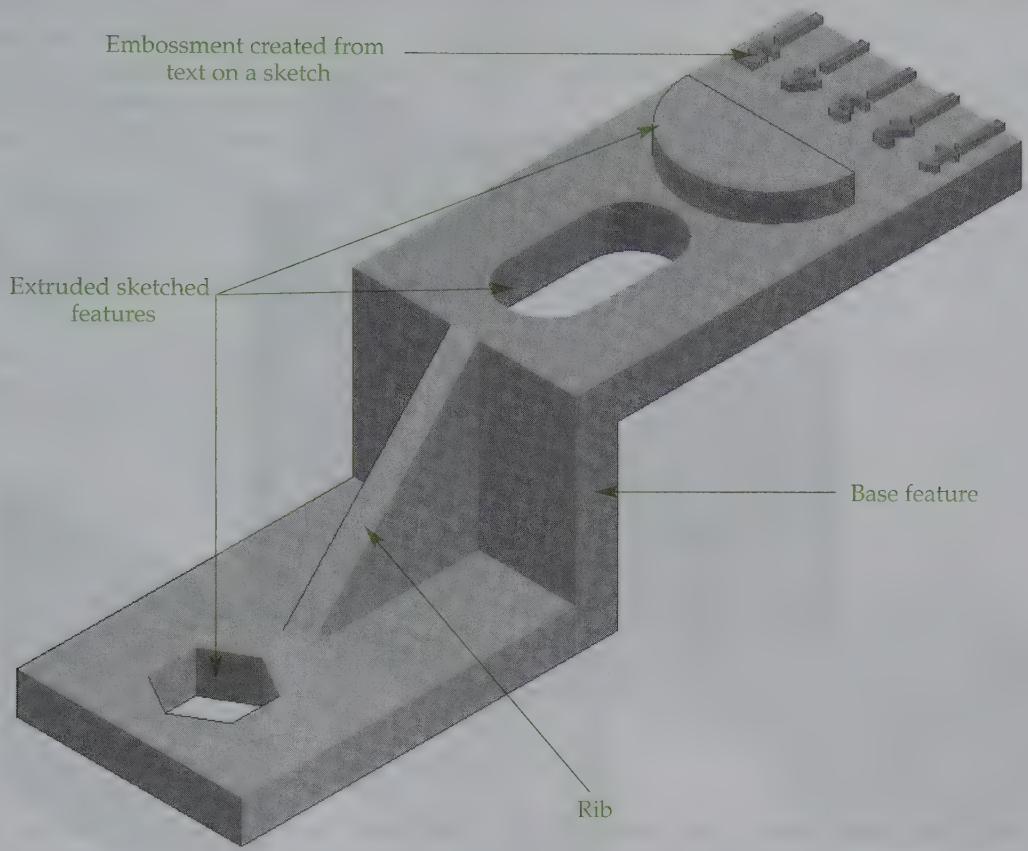


Figure 1-9.
Threads, fillets, and chamfers are a few of the placed features available in Inventor.

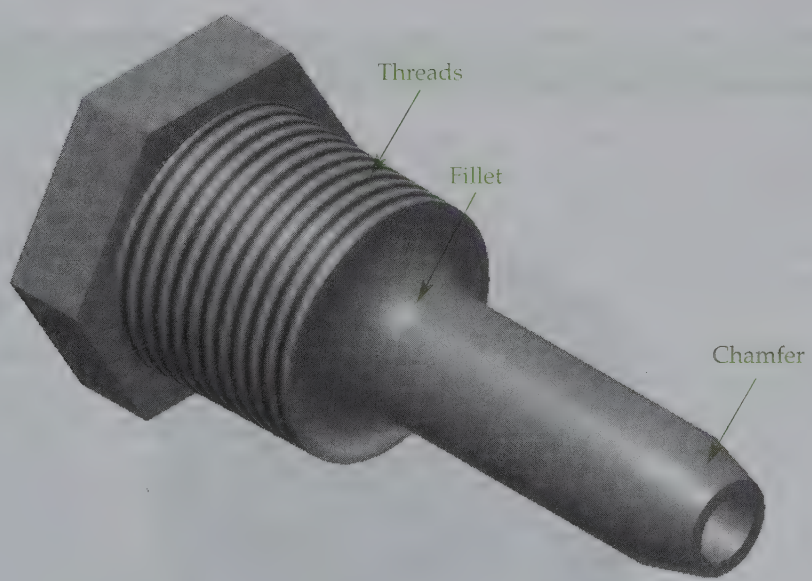
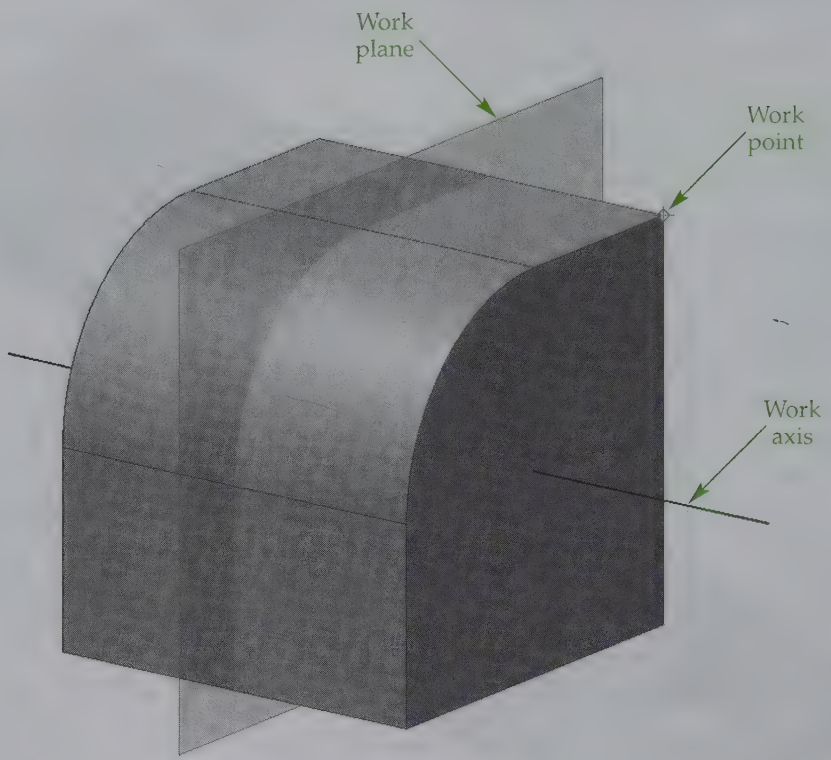
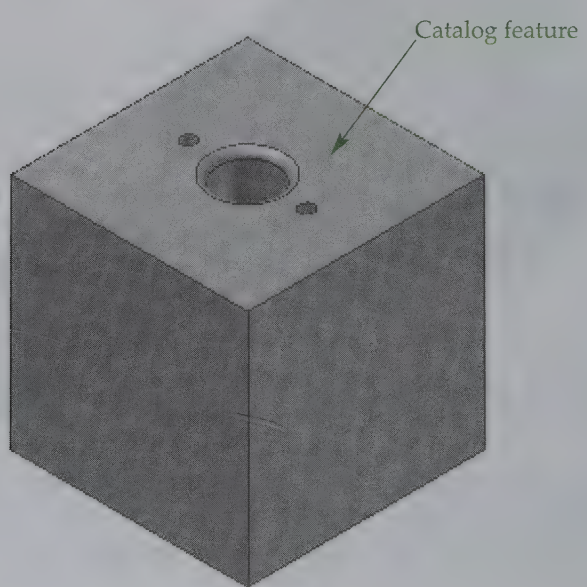


Figure 1-10.
Work features are used to build and control other features.

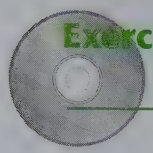


include design elements and derived parts. A ***design element***, also known as an ***iFeature***, is an existing feature or set of features you create and then save and store in a catalog to be used in other models. A ***derived component*** is more complicated than a design element and can contain a complete model consisting of several features, or even multiple parts. Derived components are often used as base features, and can be inserted into existing models.

Figure 1-11.
An example of a catalog feature, this feature contains three holes and a round.

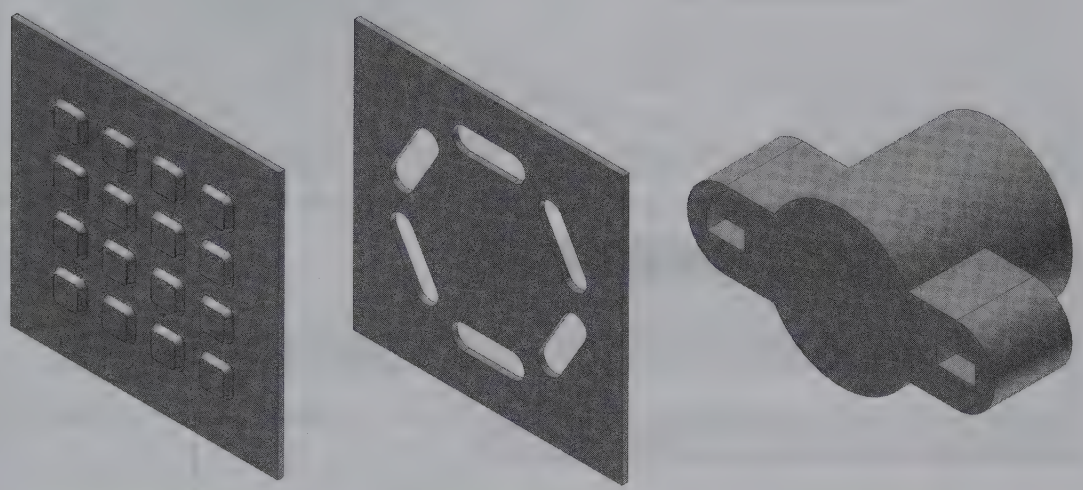


Once you create a feature, such as a hole, it may be copied into an arrangement called a ***feature pattern***. Feature patterns copy existing features, generating ***occurrences*** of the features. A ***rectangular pattern*** copies and organizes a feature into a designated number of rows and columns and places the features a specified distance apart. A ***circular pattern*** copies and organizes a feature around an imaginary circle, a designated number of times, and places each feature a specified distance from the other. A ***mirrored feature*** mirrors an existing feature over a plane. See **Figure 1-12.**



Complete the Exercise on the Student CD.

Figure 1-12.
Use patterns to create multiple copies of a feature or a group of features.



Rectangular pattern | Circular pattern | Mirrored feature

Constraints

Constraints are parameters that control the size, location, and position of model elements, including sketches and features. The use of well-defined constraints preserves specific design intensions, even as revisions are made. For example, if the center of a 10 mm diameter hole in a square plate must be 12 mm away from a specific edge, a dimensional constraint identifying this distance is added. If the edge is moved, the hole will move with it, maintaining the 12 mm distance.

Constraints define model elements and protect your model and design ideas. A model that is ***under-constrained*** has elements that are unclear, can be changed or moved, and remain undefined. See **Figure 1-13.** As you progress through the design process, you must constrain your model enough so that it becomes ***fully-constrained***, except in some adaptive situations. This ensures your model is parametric and your design is accurate.

An ***over-constrained*** model has too many dimensional or geometric constraints and generates a warning. See **Figure 1-14.** A ***modeling failure*** is the result of dimensional or geometric constraints that are impossible to apply to the model. Over-constrained and modeling failure situations must be resolved before you can create a feature and a fully parametric model.

Figure 1-13.
The hole may be moved because it is not fully constrained.

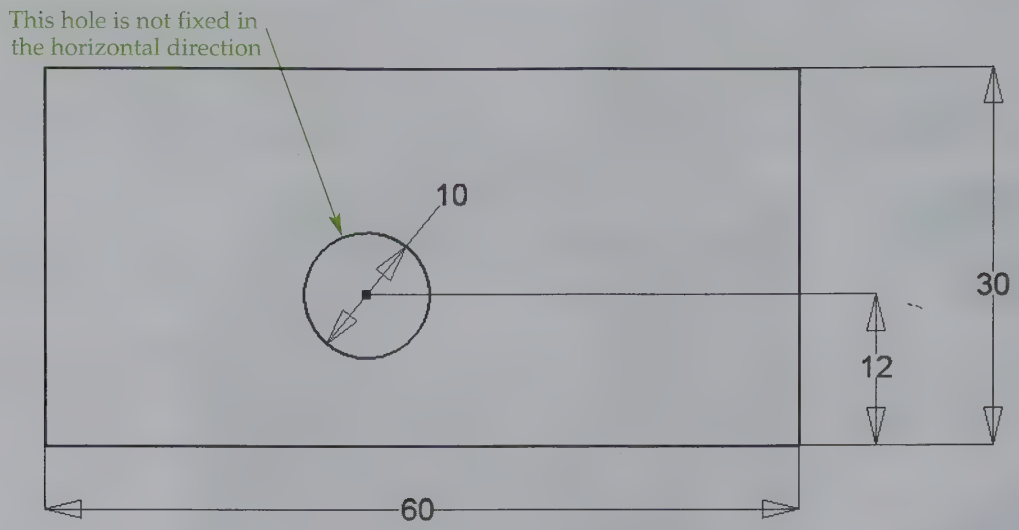
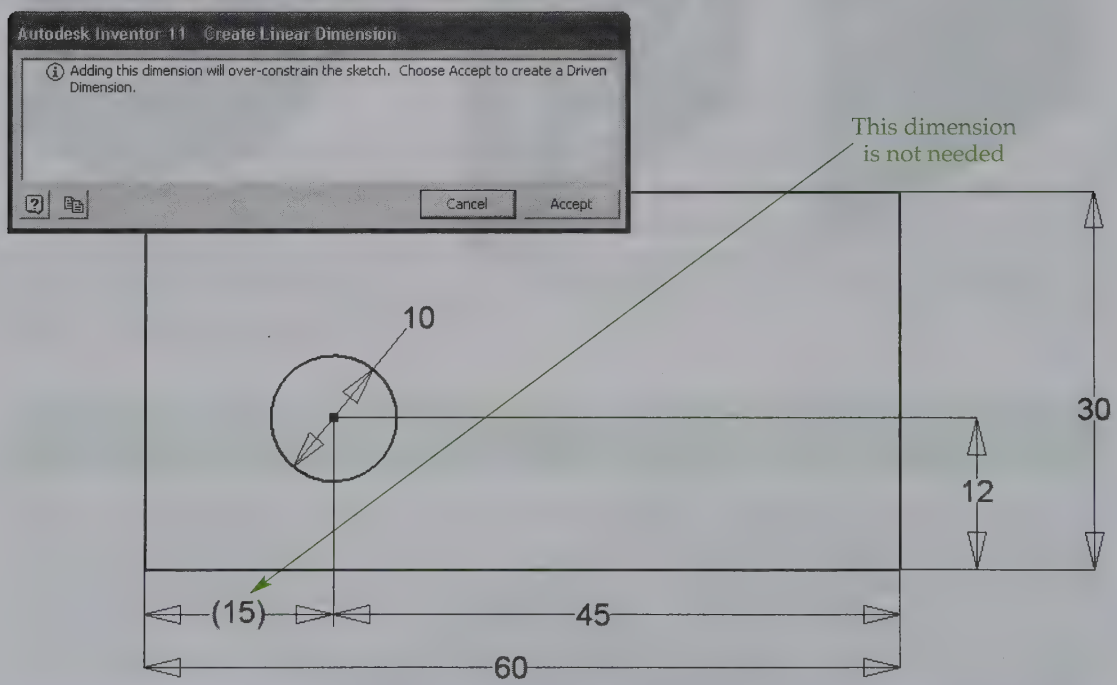


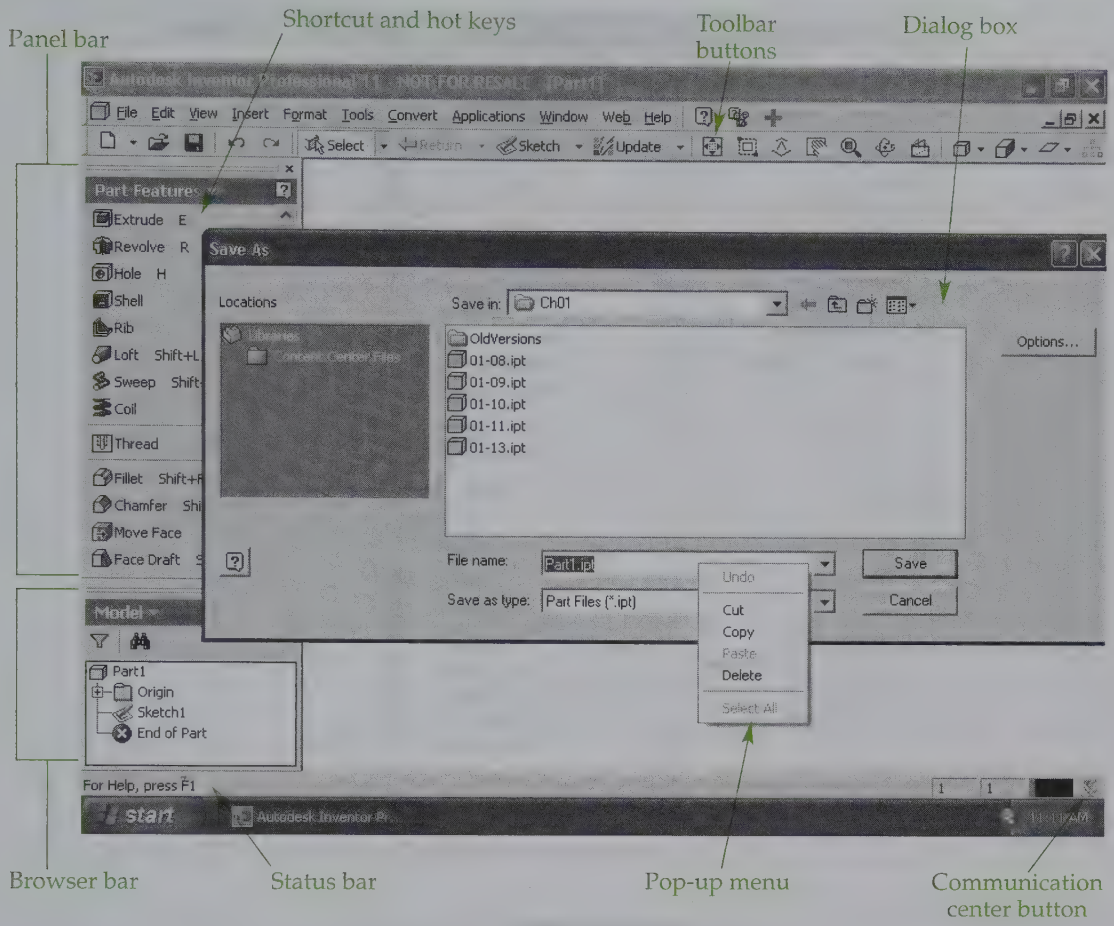
Figure 1-14.
This warning appears when you try to add a dimension that causes an over-constrained situation.



The Inventor Interface

Interface, also known as ***user-interface***, is a term describing the tools and techniques used to provide and receive information to and from a computer application. Interface items include devices such as the keyboard and mouse to input data, and the monitor to receive information. The interface also includes on-screen features, commonly referred to as the ***graphical user interface (GUI)*** available with Microsoft Windows based software programs. The Inventor graphical user interface, shown in **Figure 1-15**, includes many standard Microsoft Windows items and some specialized Inventor features.

Figure 1-15.
There are many different elements of the Inventor interface.



The Inventor interface constantly changes as you progress through a part, assembly, presentation, or drawing file. This change occurs because only the tools and options that apply to the current work environment and design or drafting task are available. Interface items such as the panel bar and pull-down menu options automatically change when you progress to different phases of the design process.

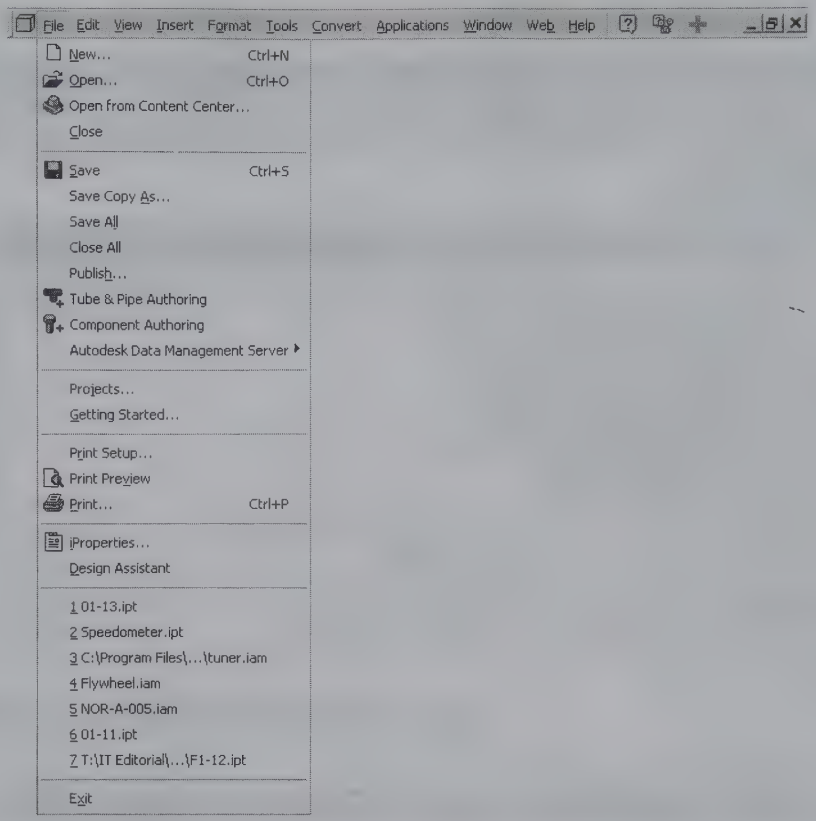
Though some interface tools and options are the same for each of the four Inventor file types, many of the options are specific to a certain file. For example, the part file interface is in part mode and contains the tools and options required to create parts. This means that only menu items, browser bar and panel bar options, toolbars, and toolbar buttons specific to parts are available. As a result, you can have different file types open at the same time, and move back and forth between different work environments.

Pull-Down Menus

The Inventor ***pull-down menu system*** provides you with a text-based, menu input configuration. The pull-down menu bar, shown in Figure 1-16, houses a number of pull-down menus that are currently available for the particular work environment. You can pick any of these pull-down menus to reveal the options for that menu and move your cursor down and up to highlight menu options.

As you select different menu options, the ***status bar***, shown in Figure 1-17, provides you with information about the tool by displaying a help string. A ***help string*** is a short description of what happens if you select the particular option over which your cursor is hovering. For example, if you highlight **File>New...**, the help string in

Figure 1-16.
The **File** pull-down menu is an example of the many pull-down menus available in Inventor.



the status bar displays: **Create a new document**. After you know which menu option you want to access, pick that option to start the tool or apply the setting.

An underlined character, known as an access key, is available for each menu bar option. ***Menu access keys*** allow you to open a menu using your keyboard by pressing the [Alt] key and the key corresponding to the underlined character. For example, you can access the **File** pull-down menu by pressing the [Alt] and [F] keys. This combination is represented in this text as [Alt]+[F]. You can then access the **Open...** option of the **File** pull-down menu, for example, by pressing the [O] key.

A pull-down menu option that displays an arrow to the right of the menu selection contains a ***cascading submenu***. A cascading submenu is a menu that appears after picking a menu selection. See Figure 1-18. To use a cascading submenu, also known as a ***hierarchical menu***, select or right-click the submenu name, or move your cursor over the submenu name, followed by picking the option you want to activate.

NOTE



To close the currently selected menu without opening another menu, press the [Esc] key, pick outside of the menu list, or pick on the menu bar. If you want to change to a different menu, move your cursor over the specified menu name or pick the menu name.

Dialog Boxes

The ***dialog box*** is a primary interface input technique used in Inventor. Most common tasks including saving a document, setting-up system variables, or opening a file usually require that you work with at least one dialog box. When you pick a pull-down

Figure 1-17.
Notice the help string that appears in the status bar when the cursor is placed over a menu option.

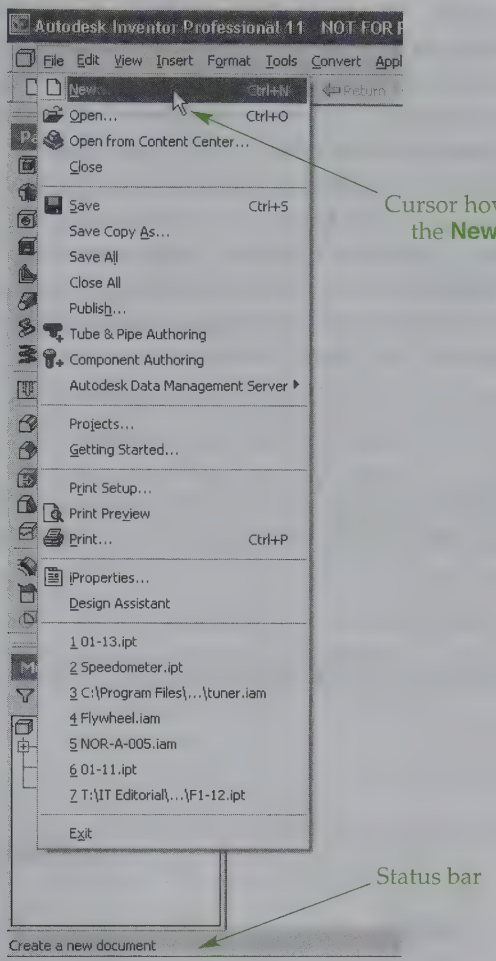
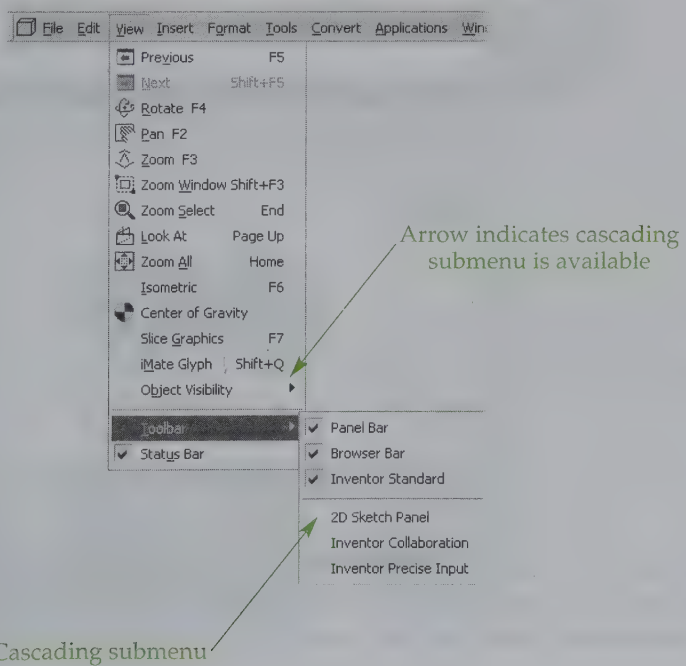


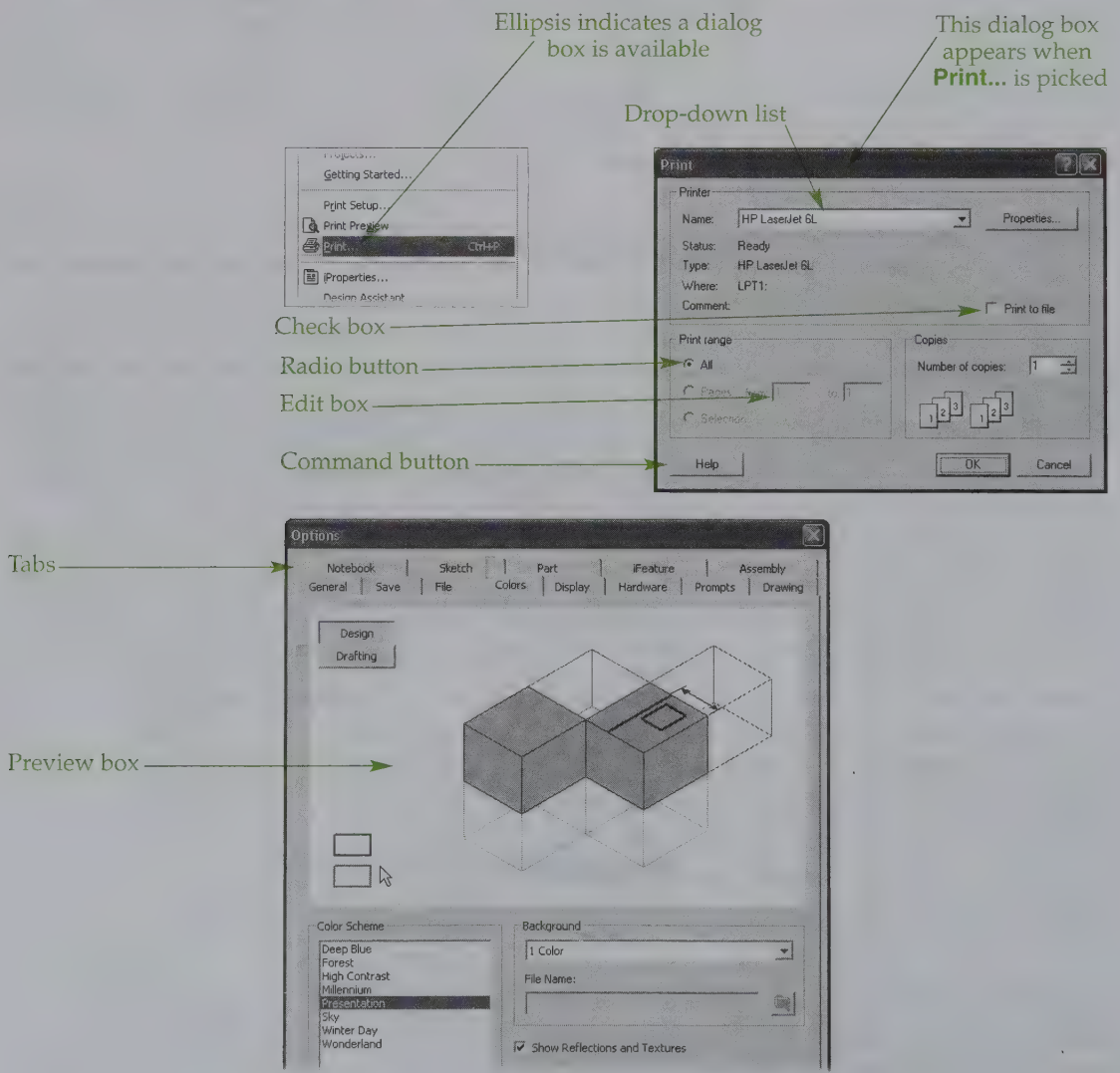
Figure 1-18.
Cascading submenus are available from menu options with a flyout arrow.



menu selection or a button that is followed by an ellipsis (...), a dialog box opens. See Figure 1-19. Most dialog boxes contain similar formats and standard items including the following:

- **Edit boxes** are used to enter a specific name or number, or a single line of information. Some edit boxes contain an up and down arrow to the right of the box that allows you to cycle through possible entries.
- **Check boxes** are used to select an option. Check boxes are filled with a check mark when they are turned on, and are empty when turned off.
- **Radio buttons** are often found in a group, and allow you to make a selection from a variety of options. Typically only one radio button can be selected.
- **List boxes**, as the name implies, display a list of possible options. Like radio buttons, typically only one option can be selected from a list box. A list box that contains a number of selections may have vertical and horizontal **scroll bars** that allow you to scroll through the possible entries. To use a scroll bar, pick the up or down arrows, or hold-down the left mouse button on the bar and drag it up or down.
- **Drop-down lists** are similar to list boxes, and allows you to choose an option from a list of possible selections. Drop-down lists differ from list boxes because

Figure 1-19.
A dialog box will appear after a menu option ending with an ellipsis is selected.



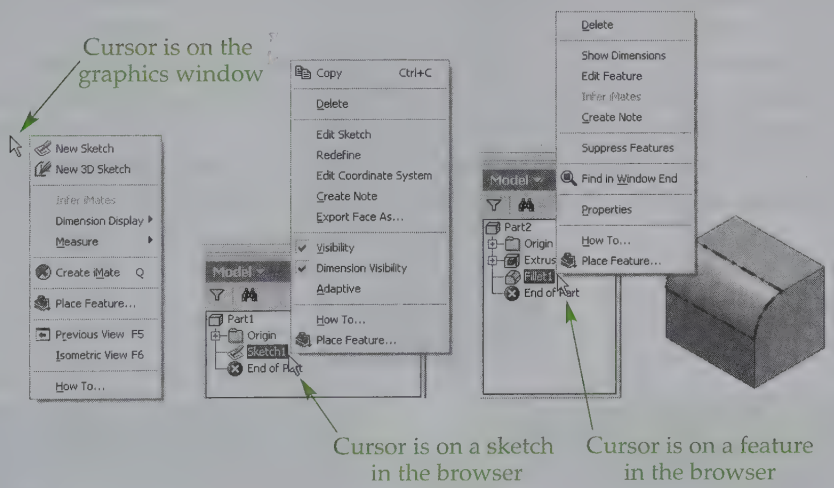
only one item is initially displayed in a drop-down list. To access more options you must pick the drop-down list or the arrow next to the box.

- **Tabs** break up dialog box content into different pages without opening additional dialog boxes or windows. Often there is so much information and so many options in a dialog box that the content is divided into different tabs. Selecting a tab displays a different dialog box page.
- **Preview boxes** or **image tiles** display an image of a selected item, so that you can recognize what you are choosing before it is actually selected and activated.
- **Command buttons** activate a specific operation, and include the **OK**, **Cancel**, and **Help** buttons found in most dialog boxes. If you pick the **Cancel** button, the dialog box closes and no changes or settings are made.

Pop-Up Menus

Pop-up menus, also know as ***shortcut*** or ***cursor menus***, are used extensively in Inventor, and allow access to tools and options by right-clicking anywhere in the graphics window or on an object or selection. See **Figure 1-20**. Pop-up menus are ***context sensitive***, which means that only items associated with the current work environment and application are available. You can access the tools and options available from pop-up menus using many of the other interface techniques. However, because pop-up menus allow you to quickly access context specific items near your cursor, you can improve the speed at which tasks are preferred, and access options that may otherwise be difficult to find.

Figure 1-20.
The type of pop-up menu that appears depends on the item under the cursor when right-clicking.



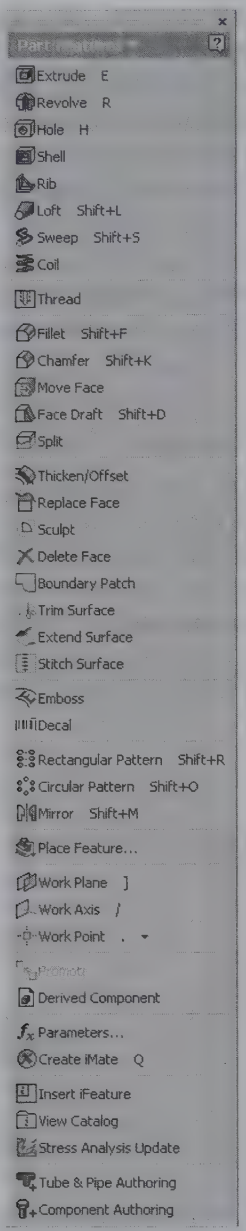
NOTE

If you right-click when a tool is active, the pop-up menu contains options specific to the tool.

Panel Bar

The ***panel bar***, shown in **Figure 1-21**, provides you with a list of several design tools available for creating sketches, models, and drawings, depending on the specific work environment. For example, only a list of options and tools used for creating parts

Figure 1-21.
The panel bar contains several design tools.

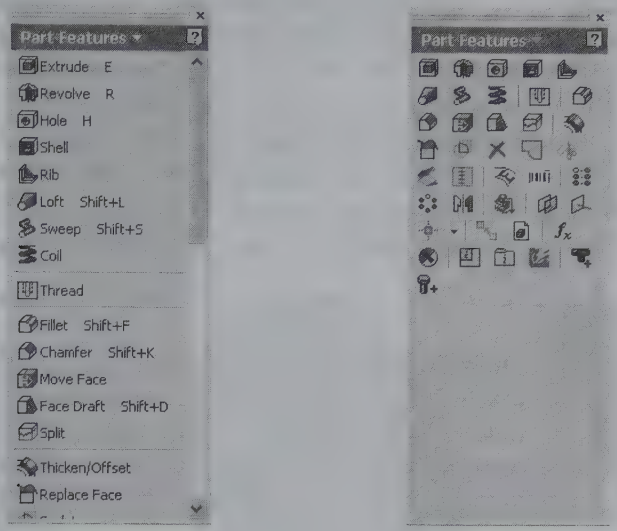


are available in the panel bar while you are working on a part. In addition, only the list of options used for a particular design stage is displayed. Inventor accomplishes this transition two ways. First, as you move from one design stage to another, the menu and tools automatically change. For example, if you are working on a part and are currently in sketch mode, only a list of sketch options is available. When you finish the sketch, the **Part Features** panel bar is displayed. Second, only tools that can be used in the current design stage are available for selection; all others are shaded.

To manually change from one list of options to another, right-click directly on, or beside, any of the tools provided in the current list of items, and select the desired panel bar mode from the pop-up menu. Another option is to click on the current panel bar name, such as **2D Sketch Panel**, and select the desired panel bar mode from the list.

Applying one of these techniques displays a list of options that includes **Expert**. See Figure 1-22. By default, the panel bar is set to ***learning mode***, which provides you with tool icons and names. Learning mode is helpful when you first begin to work

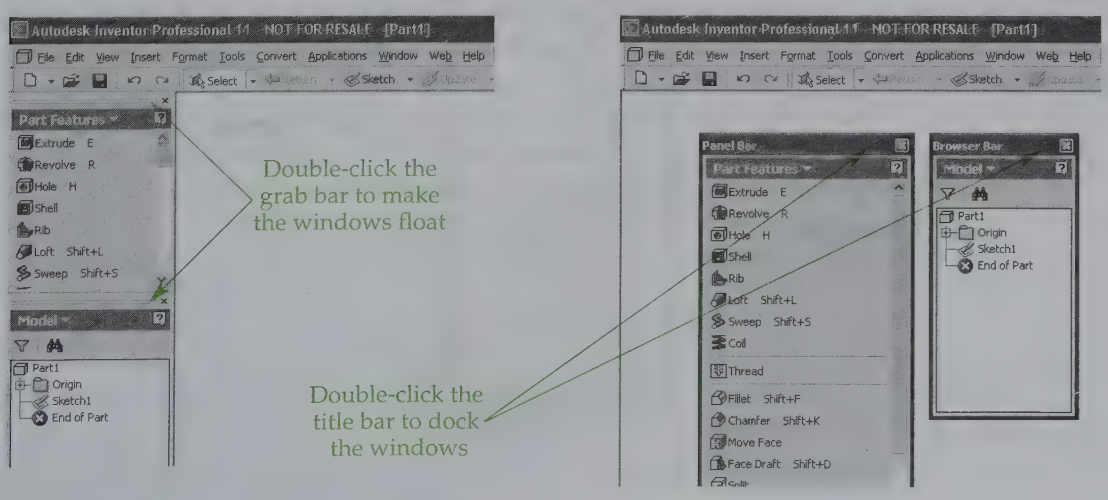
Figure 1-22.
The panel bar can be displayed with names for learning and without names to conserve screen space.



with Inventor and do not want to wait for a tooltip. Selecting the **Expert** option sets the panel bar to ***expert mode***. A panel bar in expert mode looks similar to a toolbar, and contains only tool buttons. This can be helpful if you want a smaller panel bar display allowing for a larger graphics window.

You can resize the panel bar using the resizing arrows accessible by moving your cursor over the panel bar edges. By default the panel bar is ***docked***, or locked in a position outside of the graphics window, in the upper-left corner of the work environment. To ***undock*** or ***redock*** the panel bar, double-click the ***grab bar***, the area at the top of the docked panel containing two thin bars. You can also pick the ***grab bar*** and drag the panel bar to a new location. When a window is undocked, it is considered to be ***floating***. A floating window can be placed almost anywhere on the screen. If this window is moved to a location where it can be docked, an outline of the window will move to the docked position. See Figure 1-23.

Figure 1-23.
You may prefer to have some windows docked and others floating.



Browser Bar

The ***browser bar***, or ***browser***, displays all the items in your model or drawing. See Figure 1-24. The number and type of items available in the browser varies depending on the current design stage, model type, and work environment. Typically, the name of the file and an icon is displayed on top, followed by the **Origin** folder and all the origin options, which is then followed by sketches, features, or file components. Items in the browser are listed in the order they were created or inserted, though it may be possible in some applications to drag and drop browser items up or down in the list. Finally, **End of Part** displays last, unless moved. For example, Figure 1-25 shows a completed part and how dragging and dropping **End of Part** changes the display.

PROFESSIONAL TIP



Drag **End of Part** above an existing feature, or drag features below **End of Part**, anytime you want to add a new sketch or feature to a model at an earlier point in the design, or just to observe the design history.

Many of the items displayed in the browser are arranged in what is known as a ***tree structure***, which contains parent and child nodes. A ***parent node*** is associated with the main item, such as a feature that contains a sketch. The **Origin** folder is another example of a parent node. A parent node has ***child nodes***. See Figure 1-24. Child nodes are sketches, features, or components that create the main item. To display child nodes, pick the **Expand** button to the left of the object name or right-click on the object name and select the **Expand All Children** pop-up menu option. To hide child nodes, pick the **Collapse** button to the left of the object name or right-click on the object name and select the **Collapse All Children** pop-up menu option.

You can resize the browser bar using the resizing arrows accessible by moving your cursor over the panel bar edges. By default the browser bar is docked in the upper

Figure 1-24.
The browser bar contains information regarding the objects in a model or drawing.

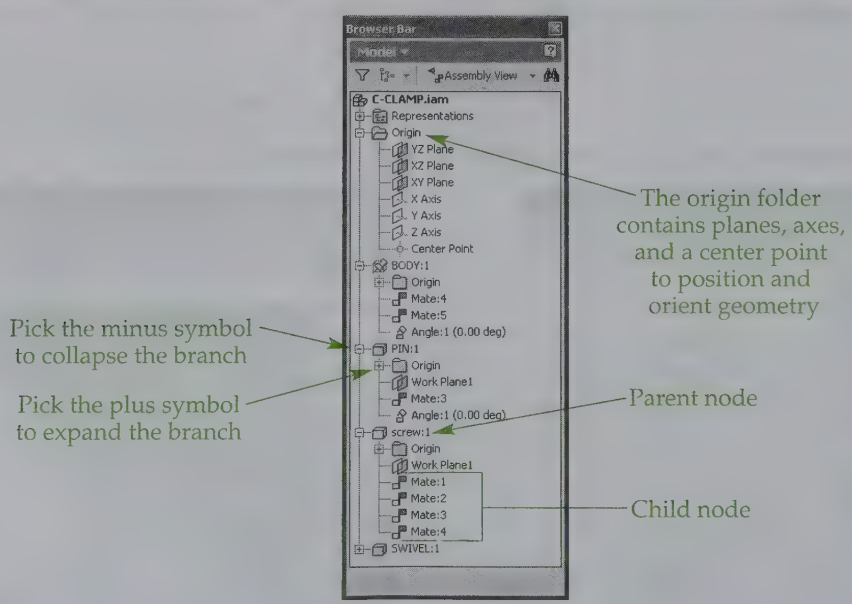
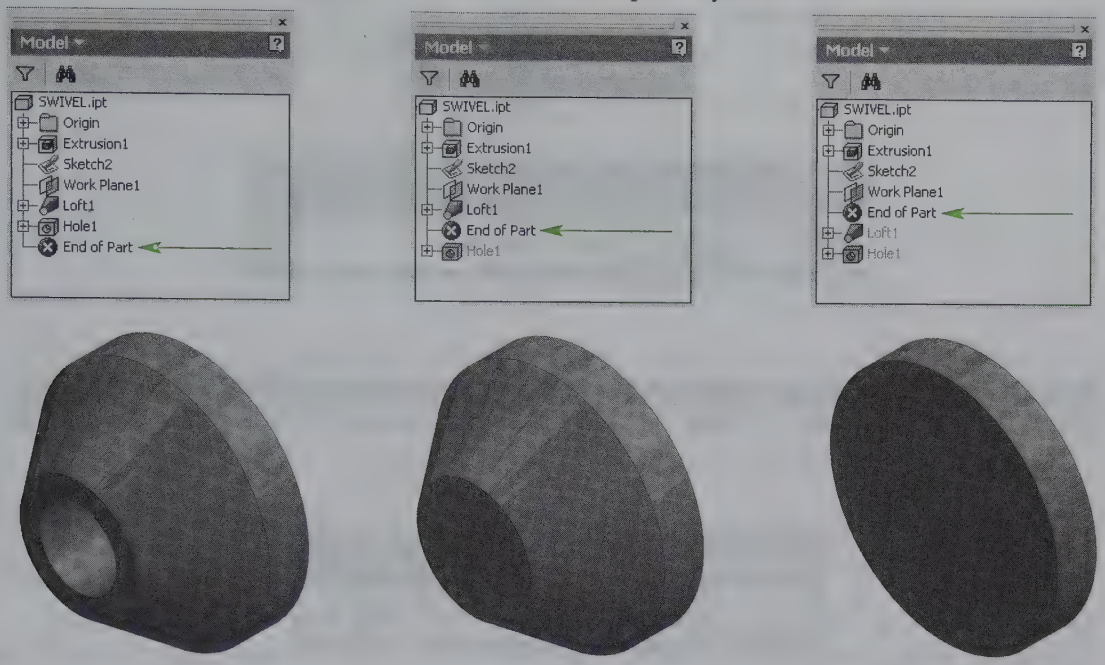


Figure 1-25.
Moving the **End of Part** indicator in the browser temporarily turns off items in the model.



left corner of the work environment. To undock or redock the browser bar, double-click above the browser bar name, or pick the grab bar above the name and drag the browser bar to a new location. The grab bar is represented by two thin bars at the top of the docked panel.

Toolbars

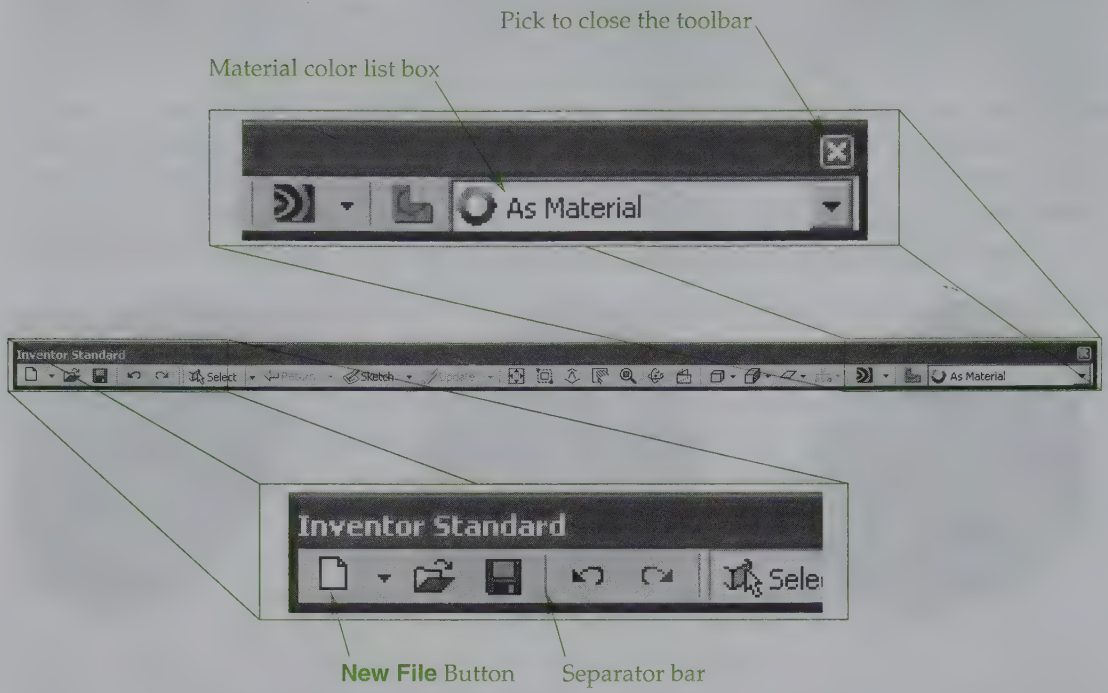
Toolbars are important interface elements that display several ***toolbar buttons***, each with a specific icon, that allow access to Inventor tools and variables. Toolbars have thin lines that separate groups of similar tools, known as ***tool groups***. There are a number of toolbars that apply to specific tasks, but by default, the first time you start Inventor, only the **Inventor Standard** toolbar is shown. See Figure 1-26. Toolbars have features similar to pull-down menus and the panel bar, but usually require you to select only one toolbar button and take up less space on your screen.

Most Inventor tools are contained in at least one toolbar. For example, the 2D sketch **Line** tool is located in the **2D Sketch Panel** and the **2D Sketch Line** toolbars. Some toolbars are available by default by picking **View>Toolbar** or from the pop-up menu displayed when you right-click on an existing toolbar. However, to access most toolbars, you must customize the work environment settings, using the **Customize** dialog box. The easiest way to make a specific toolbar available is to:

1. Select **Customize...** from the **Tools** pull-down menu or right-click on any existing toolbar and pick the **Customize...** pop-up menu option.
2. Select the **Toolbars** tab of the **Customize** dialog box.
3. Choose the desired toolbar from the **Toolbars:** list box.
4. Pick the **Show** button.
5. Pick the **Close** button to exit the **Customize** dialog box.

You can resize toolbars using the resizing arrows accessible by moving your cursor over toolbar edges. To undock or redock toolbars, double-click an area on the toolbar where there is no tool button, or pick an area where there is no tool button and drag the toolbar to a new location. A toolbar grab bar is represented by one thin bar on the left side of the docked toolbar.

Figure 1-26.
These detail images of the **Inventor Standard** toolbar show the different types of features that may be found on all toolbars.



Inventor toolbars have tooltips and help strings that help you understand and recognize different toolbar buttons. *A **tooltip*** is a small text box that pops-up under the toolbar button when you point to the button with your cursor, and displays the tool button name. *A **help string***, which describes the button function, appears in the status bar when the tooltip pops-up.

Figure 1-27 shows the tooltip for the **Open** button on the **Inventor Standard** toolbar. The tooltip tells you that the icon you see is the **Open** tool. The help string is displayed on the status bar and tells you the function of the **Open** button: **Open an existing document**. Once you review the toolbar button icon, see the tooltip, and read the help string, pick the button to activate the tool.

Some toolbar buttons have an arrow to the right of the button, known as a flyout. See Figure 1-28. A ***flyout*** is a button that contains additional, related tool buttons, much like a cascading submenu. To use a flyout toolbar button, select the arrow next to the button. This opens the other options associated with the specified button. These buttons work exactly the same as other toolbar buttons and provide tooltips and help strings.

Figure 1-27.
Place the cursor on an icon to reveal the icon's name as a tooltip.

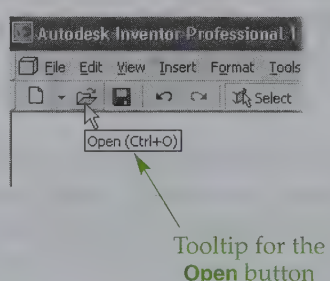
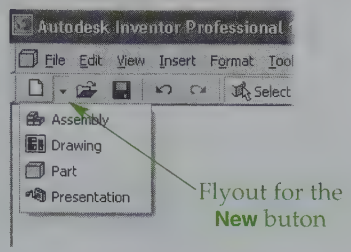


Figure 1-28.
Pick the flyout, next to some icons, to see additional tool buttons.



NOTE

Most of the time, when you pick a flyout option, the selected button becomes active and is displayed on the toolbar on the top level of the flyout, hiding the rest of the buttons.

Shortcut and Hot Keys

Inventor ***shortcut keys*** allow you to initiate certain predefined tools by pressing two keys. Inventor ***hot keys*** allow you to access certain predefined tools by pressing a single key. Like many of the Inventor interface components, particular shortcut and hot keys are available only while in a certain work environment. Some of the options in certain pull-down menus, such as selections in the **File** and **Edit** pull-down menus, have a shortcut key or hot key displayed to the right of the option name that allow you to access certain predefined tools using your keyboard, without opening the pull-down menu. **Refer to Appendix A for a complete list of Inventor shortcut and hot keys.**

NOTE

Shortcut keys that require you to press the [Ctrl] key and an additional key, such as [Ctrl] and [O] ([Ctrl]+[O]), are referred to as ***control keys***.

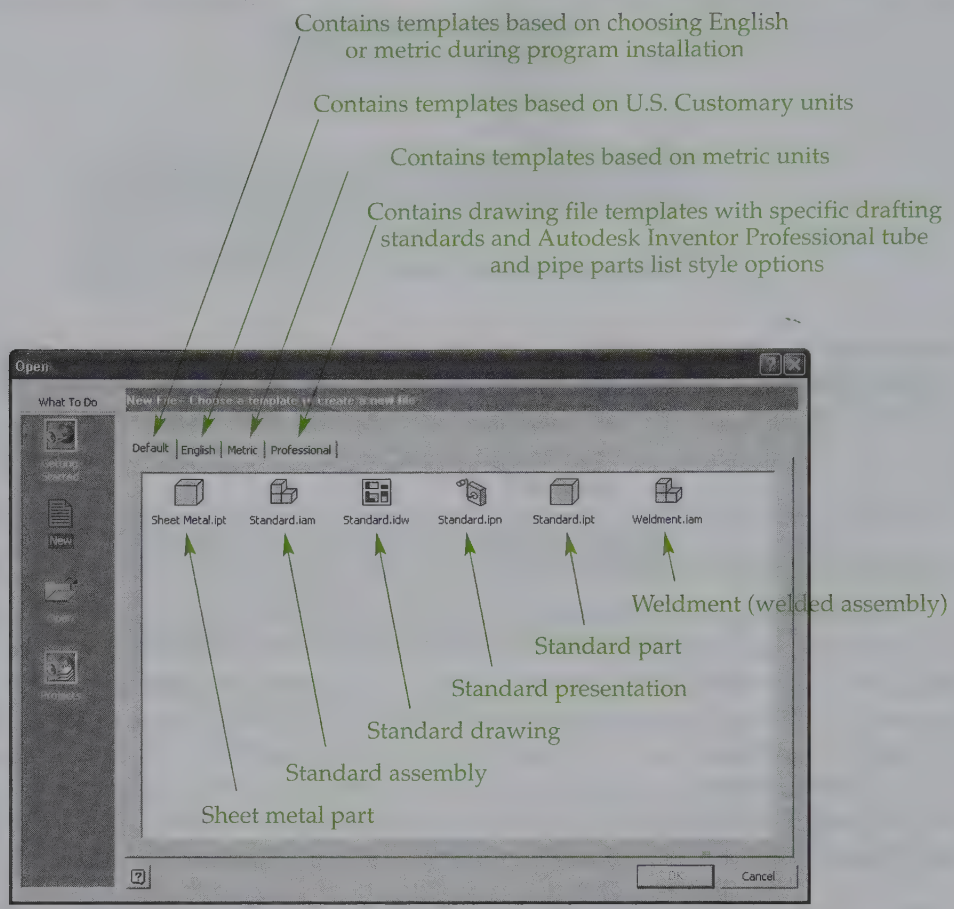
Starting a New File

The **New File** window of the **Open** dialog box, shown in Figure 1-29, is the primary tool used to access templates for creating new Inventor parts, assemblies, presentations, and drawings. A ***template*** is a file with predefined settings, often also referred to as a ***prototype***. Display the **New File** window of the **Open** dialog box by picking **File>New...**; pressing [Ctrl] + [N]; or picking the **New** button on the **Inventor Standard** toolbar.

NOTE

The **New** button on the **Inventor Standard** toolbar, shown in Figure 1-28, is flyout that contains **Assembly**, **Drawing**, **Part**, and **Presentation** options. These selections can be used to quickly access a new standard part, assembly, drawing, or presentation template file. The templates located in the **New** flyout button are those found in the **Templates** folder and listed in the **Default** tab of the **Open** dialog box, **New File** window.

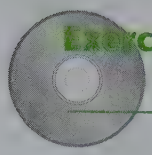
Figure 1-29.
Pick the various tabs to see the many templates available in the **New File** window.



Initially, the **New File** window contains **Default**, **English**, **Metric**, and **Professional** tabs. Inside each tab are Inventor template files. As shown in Figure 1-29, the four Inventor file types are represented by icons. You can specify the **Default** tab as **English** or **Metric** when you install the program, based on the units you typically use, and which you would like to designate as default. The **English** and **Metric** tabs hold specific unit templates for each Inventor file type including ANSI in the **English** tab, and ANSI, BSI, DIN, GB, ISO, and JIS in the **Metric** tab.

Sheet metal and weldment templates are also available in the **New File** window. These templates are not actually separate file types, but more like subfiles preset to be used for specific model applications. Sheet metal part templates are similar to other part templates and carry the same file extension, .ipt. Opening a new Sheet Metal.ipt file from the **New File** window creates a part as a sheet metal subfile and opens the sheet metal tools necessary to create sheet metal parts. Weldment templates are similar to other assembly templates and carry the same file extension, .iam. A ***weldment*** is an assembly in which parts are fixed together with welds. Opening a new Weldment.iam file from the **New File** menu creates an assembly as a weldment subfile and opens the welding tools necessary to create weldments. *A* ***weldment*** is an assembly in which parts are fixed together with welds.

To begin a new file using the **New File** window of the **Open** dialog box, double-click the desired file type icon, or pick the file and the **OK** button in the lower right-portion of the **Open** dialog box.



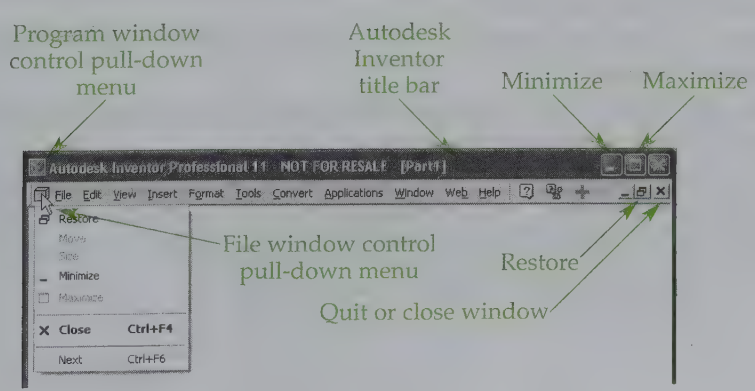
Complete the Exercise on the Student CD.

Controlling Windows

The **Window Control** pull-down menu and the corresponding window control buttons, shown in Figure 1-30, are used to help you organize Inventor program and file windows. The **Window Control** pull-down menu for Inventor is accessed by picking the **Inventor** icon in the upper left corner of the program window. The following options are displayed when you pick the **Inventor** icon:

- **Restore.** This option is available when the window is ***maximized***, which means that the Inventor window is consuming the entire extents of the screen display. Select **Restore** to restore the window to a format in which you can control the size and location of the window. You can also restore a window by selecting the **Restore** button in the upper right corner of the program window.
- **Move.** This selection is available when the window has been restored and allows you to move the window to a new location by dragging the Inventor title bar. However, you can move a restored window by dragging the Inventor title bar, without selecting the **Move** option.
- **Size.** This option is available when the window has been restored and allows you to resize the window by dragging any of the four window edges or corners to a new location. However, you can resize a restored window by dragging any of the four window edges or corners, without picking the **Size** option.
- **Minimize.** This selection is available when the window is in a restored or maximized format. Pick **Minimize** to fully reduce the size of the window, without closing the program. You can also minimize a window by picking the **Minimize** button in the upper right corner of the program window.
- **Maximize.** This option is available when the window is in a restored or minimized format. Pick **Maximize** to fill the extents of your screen with the program window. You can also maximize a window by selecting the **Maximize** button in the upper right corner of the program window.
- **Close.** Pick this option to close all files and completely exit Inventor. You can also close Inventor by picking **File**>**Exit**.

Figure 1-30.
These window controls are typical in a Microsoft Windows® environment.



NOTE

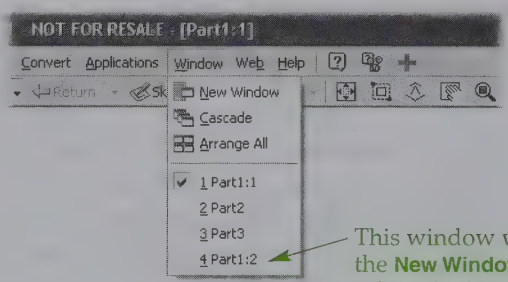
When you close a file using one of the **Close** tools, or the Exit tool, several alert boxes can appear depending on the situation. For example, if you have not saved your work, you are asked if you want the file to be saved before closing.

Inventor allows for a ***multiple document interface***, also known as ***multiple design interface***, which means that you can have several documents or document views open at the same time. You can use individual file **Window Control** pull-down menus, and the **Windows** pull-down menu options to manage the multiple document interface. The **Window Control** pull-down menu for an Inventor file is accessed by picking the corresponding Inventor file icon. If the file window is maximized, the **Window Control** pull-down menu is located to the farthest left of the pull-down menu bar. However, if the file window is restored or minimized, the **Window Control** pull-down menu is located to the left of the file menu bar, in the graphics window. The menu options available when you select a file icon function much like program window control menu options. However, when you restore, move, size, minimize, maximize, or close a file window, the operation affects only the file window, not the entire program. Additionally, the **Next** menu option is available, which allows you to toggle between multiple opened files.

The **Window** pull-down menu, shown in Figure 1-31, contains the following options that can be used to control the multiple document interface:

- **New Window.** This option allows you to create a new window for the active document. Additional windows enable you to have several design views

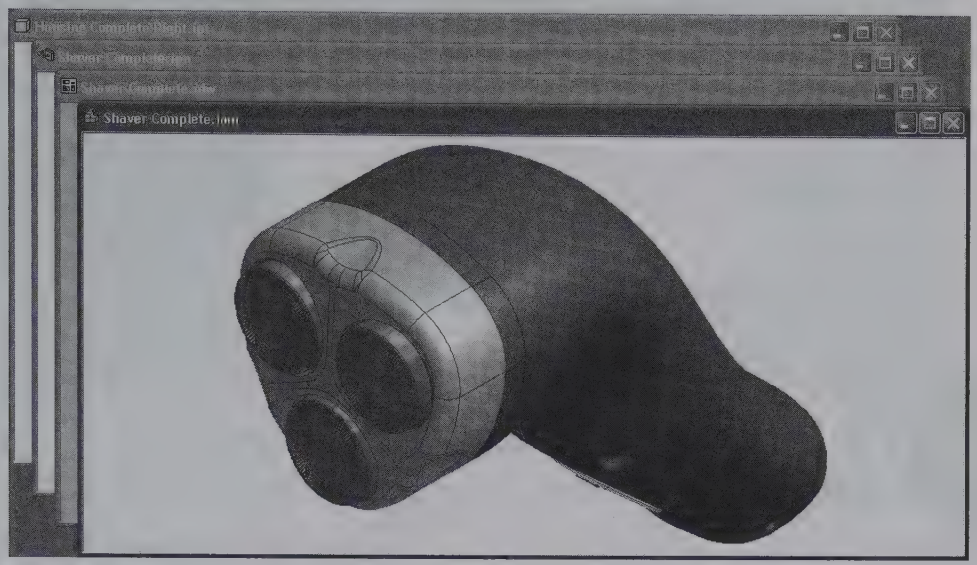
Figure 1-31.
Use the **Window** pull-down menu to create a new window, position the current windows, and view the list of open files.



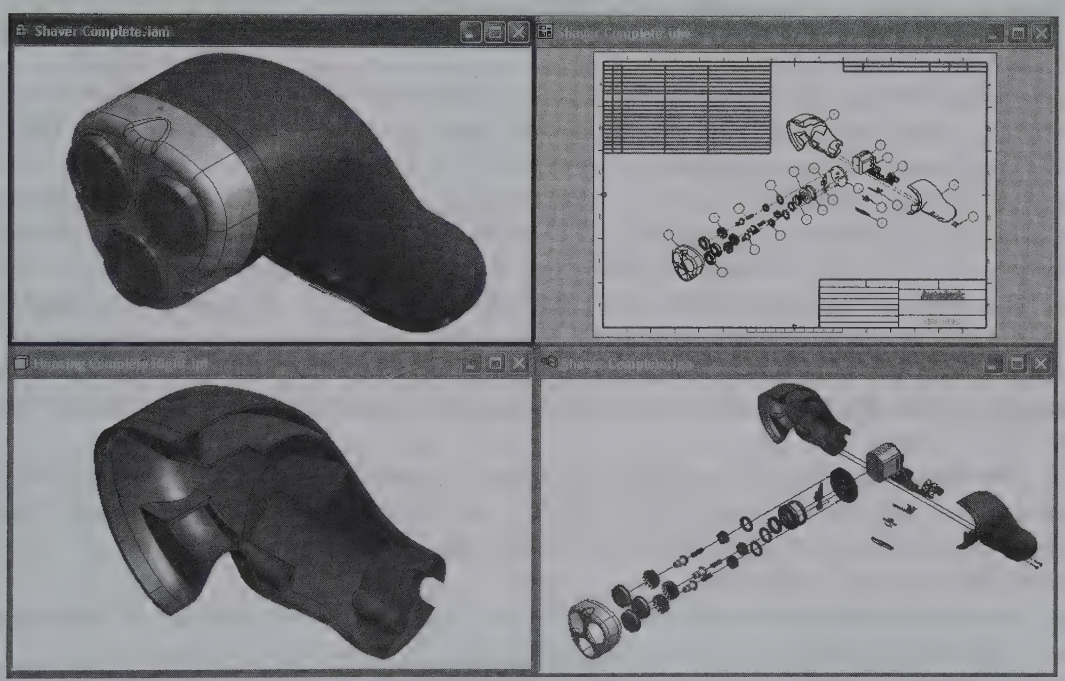
opened and accessible at once.

- **Cascade.** This option arranges all document windows in a cascading fashion, as shown in Figure 1-32.
- **Arrange All.** This option arranges all document windows in a way that shows the entire window of each. See Figure 1-32.
- **Multiple document access.** The names of all currently opened documents are displayed at the bottom of the **Window** pull-down menu. See Figure 1-33. The file name with a check mark next to it is currently the active window. To make a window active, pick the name.

Figure 1-32.
These are examples of window positioning when the **Cascade** and **Arrange All** options are selected from the **Window** pull-down menu.



Cascade

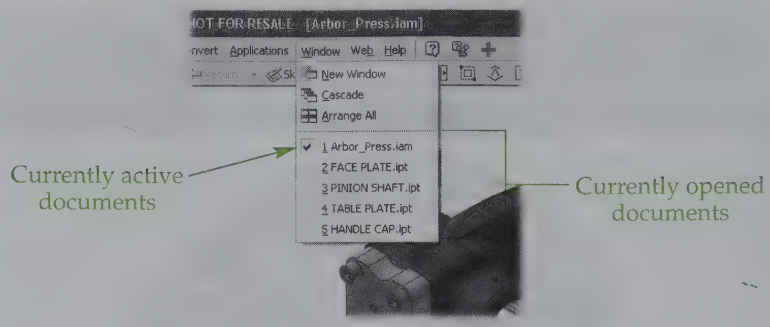


Arrange All

Note

Each window available in a document can be minimized, maximized, closed, and manipulated in size. Files can also be closed by picking **File>Close**. If you have multiple documents opened, you can close them all together by picking **File>Close All** or exiting Inventor.

Figure 1-33.
The **Window** pull-down menu contains the names of the open files and a checkmark next to the active file.





Exercise 1-4 Complete the Exercise on the Student CD.

Saving Your Work

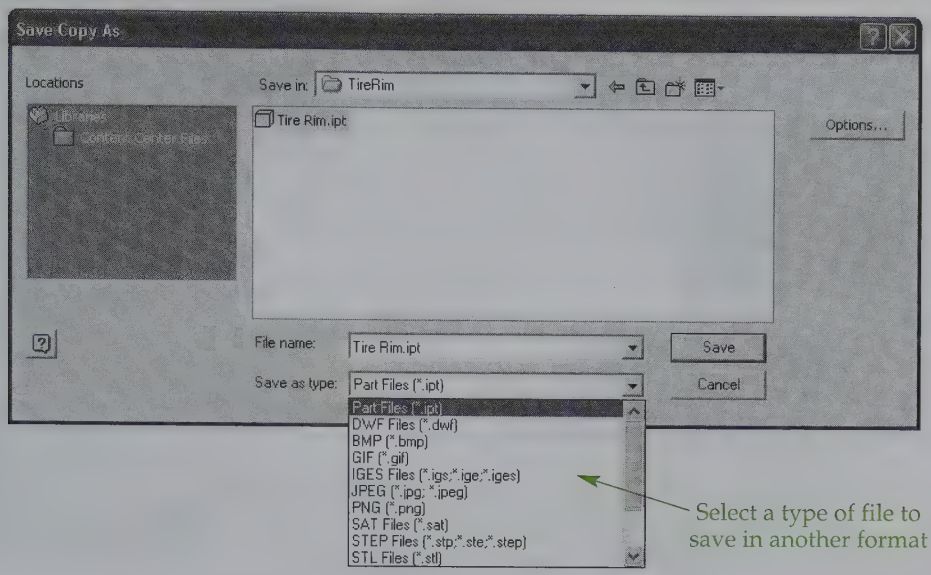
After starting a model or drawing, you need to save the file. A sudden loss of power or a computer problem may cause the loss of your file, unless it is saved. This is why you should save your work frequently. In fact, you may want to save your work every five to ten minutes. To access the **Save** tool, pick the **Save** button on the **Standard** toolbar, pick **File>Save**, or press [Ctrl]+[S]. The **Save** tool resaves the file if it has already been saved, or opens the **Save As** dialog box if the file has not been previously saved.

You may need to save the currently open file under a new file name. For example, a part you are developing is similar to an existing part. Open the existing part and save it under the new name, close the old part, and open the new part. This is done using the **Save Copy As...** command. To access this command, pick **File>Save Copy As...** See Figure 1-34.

The **Save Copy As...** command is also used to save a file in a format that is not a standard Inventor format, such as BMP, JPEG, or STEP. These file formats may be necessary when collaborating with other individuals. The following options are located inside the **Save As** dialog box:

- **Locations.** This list box shows a list of the file paths in reference to the active project. You can select a path to display the files and subfolders.
- **Save in:** Use this drop-down list to navigate through drives and folders to find a certain file by selecting the down arrow next to the drop-down list box. The list box is located below the **Save in:** drop-down list and shows the names of the files located in the current directory.
- **Last.** This button, to the right of the down arrow, allows you to go back to the last folder visited.
- **Up One Level.** This button allows you to return to the menu level of the previously opened, or parent, folder.
- **Create New Folder.** This button allows you to create a new folder if desired. Locate the specific area where you want a new file, and select the **Create New**

Figure 1-34.
You can save a file as another file type in the **Save Copy As** dialog box.



Folder button. Now, type a folder name over the existing New Folder default name. Finally, pick the folder you just created or press [Enter] to open the folder.

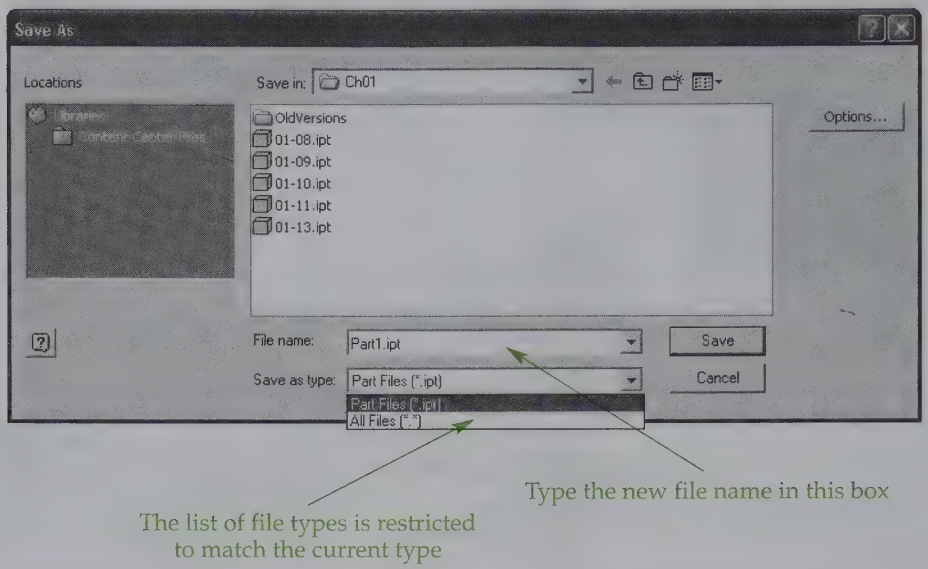
- **View Menu.** This flyout button allows you to view drives and folders in one of the following formats: large icons, small icons, list, details, or thumbnails.
- **File name:** This edit box shows the currently selected and highlighted Inventor file. To display the file name, pick the file from the list box or type the name of the file in the **File name:** edit box.
- **Save as type:** This drop-down list reveals the difference between the **Save As** and the **Save Copy As** dialog boxes. The format of the original file (*.ipt, *.ide, *.iam, *.idw, or *.ipn) is available in the **Save As** dialog box. This drop-down list in the **Save Copy As** dialog box contains various formats for saving (*.dwf, *.bmp, *.gif, *.iges, *.jpeg, *.png, *.sat, *.step, *.stl, *. tiff, *.xgl, and *.zgl).
- **Options...** Picking this button opens the **File Save Options** dialog box. The **File Save Options** dialog box controls the method of generating a preview image for the file. This image is displayed in the Preview area of the **File Open** dialog box.

Many design projects require that multiple files be open. If you make changes to these files and would like to save all of them, use the **Save All** command so each file does not have to be saved separately. To access the **Save All** command, pick **File>Save All**. If the files were saved in the past, **Save All** resaves the file. If they were not saved, the **Save As** dialog box appears. Saving a file for the first time causes the **Save As** dialog box to appear. This dialog box presents the same options as the **Save Copy As...** dialog box, except for the number of formats available. See Figure 1-35.



Complete the Exercise on the Student CD.

Figure 1-35.
The **Save As** dialog box appears if the current file has not been saved.



Opening Existing Files

The **Open File** window of the **Open** dialog box, shown in Figure 1-36, is the primary tool used to access existing Inventor part, assembly, presentation, and drawing files. Display the **Open** dialog box by picking **File>Open**; pressing [Ctrl]+[O]; or picking the **Open** button on the **Standard** toolbar.

NOTE

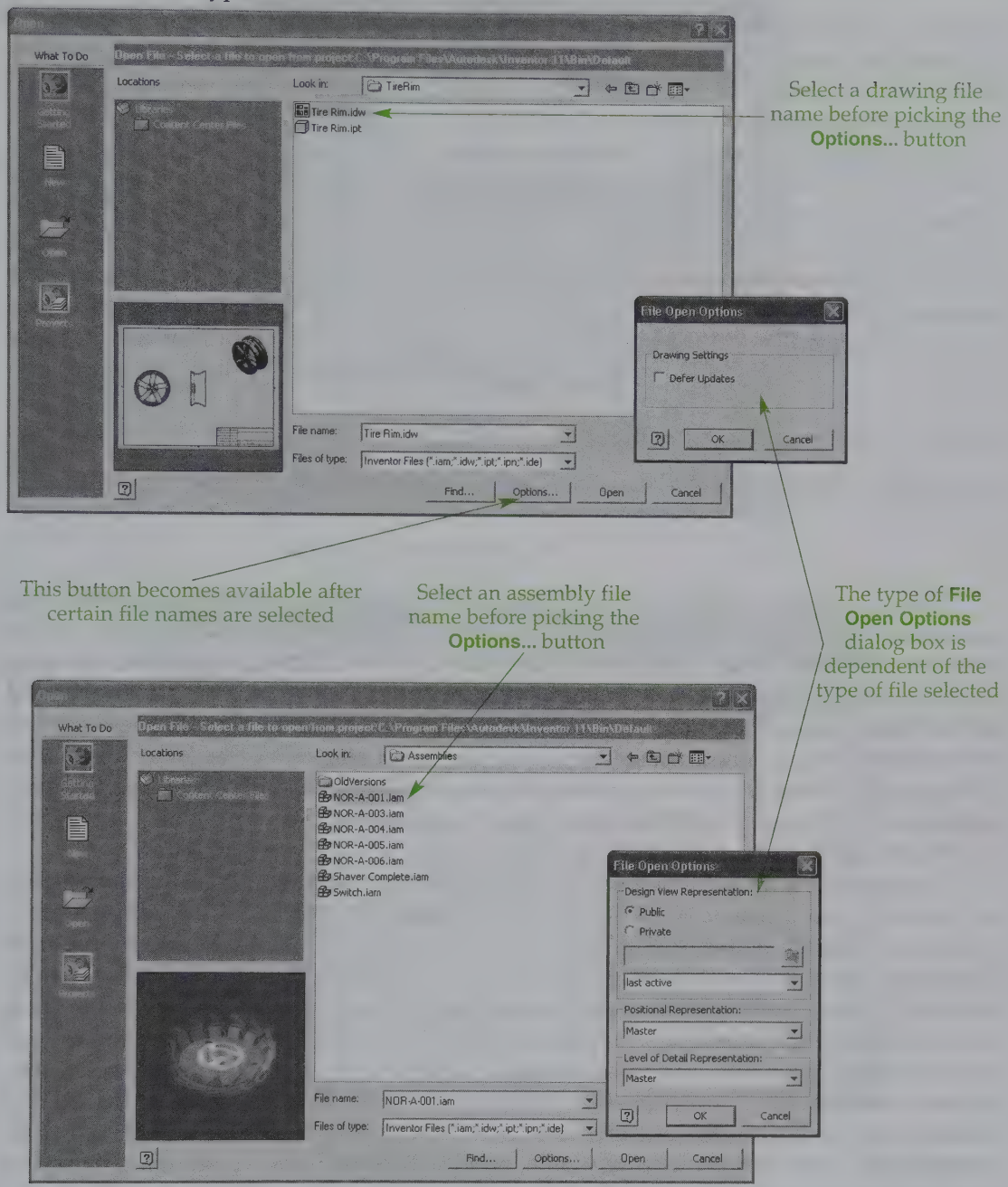


You can also open existing files directly from Microsoft Windows Explorer, or you may be able to find files using the **Quick document** access area of the **File** pull-down menu. The **Quick document** access area is presented near the bottom of the **File** pull-down menu, and lists the path and file name of several previously used documents. This option is helpful if you are constantly opening the same recently used files. Each of the paths is associated with a number. Number **1** is the most recently opened, while the highest number is the file opened longest ago.

When you access the **Open File** window, the location folder assigned to the active project is automatically displayed allowing you to quickly access files associated with the active project. However, you can navigate to any files on your computer or the network using the **Open File** window, files do not have to be in the active project. The options in the **Open** dialog box are similar to those found in the **Save** and **Save Copy As...** dialog boxes.

Once a file is selected, the **Options...** button may be made available. Picking this button opens the **File Open Options** dialog box. The choices available in this dialog box vary depending on the type of file selected. In general, if you pick a drawing file the **File Open Options** dialog box contains a check box that allows you to defer updates, saving file load time. If you select an assembly file, the **File Open Options** dialog box contains design view and positional representation options. See Figure 1-36.

Figure 1-36.
After picking a file in the **Open File** window, pick the **Options...** button to see options that are specific to the type of file selected.



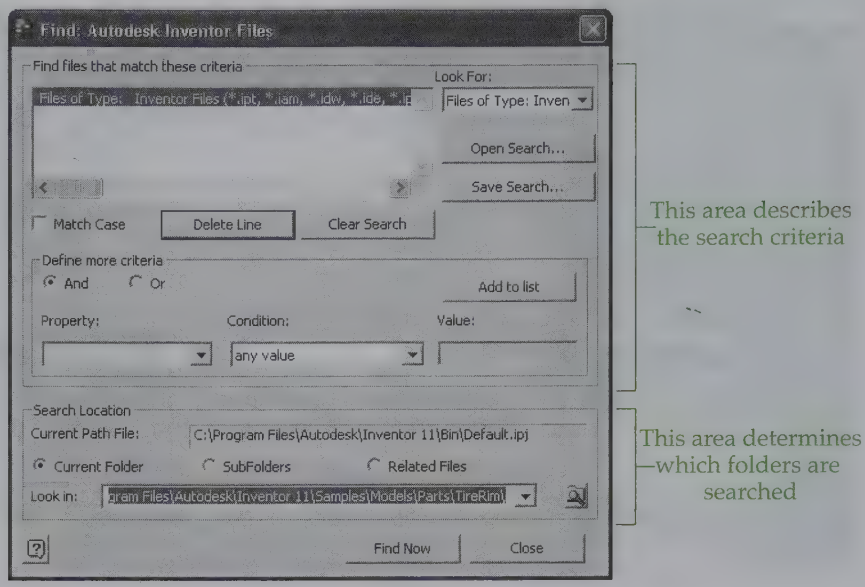
NOTE

Double-clicking an Inventor file found in Microsoft Windows Explorer should cause Inventor to start, if not already running, and the selected file to open.

Finding Files

If you forget where an Inventor file has been placed, Windows has a search tool that you may be familiar with, but Inventor also has a method of finding files that may be quicker to use. Inventor's search tool is conveniently located in the **Open File** window of the **Open** dialog box. Pick the **Find...** button to open the **Find: Inventor Files** dialog box. See **Figure 1-37.**

Figure 1-37.
Use criteria specification in the **Find: Inventor Files** dialog box to locate files.



The upper region of this window is labeled **Find files that match these criteria**. This area describes the search that is currently specified. The default search displays in the window all Inventor files (*.ipt, *.ide, *.iam, *.idw, *.ipn). Picking **Delete Line** will erase all of the contents in the window, including the default search. You can return the default line and reset all search options by picking **Clear Search**. Select the **Match Case** checkbox to find files with the same uppercase or lowercase text defined in the search parameters. If the search criteria will be used repeatedly, use the **Save Search...** and **Open Search...** buttons to save time. The **Define more criteria** area enables you to develop additional search criteria by constraining your search for a file.

The **Search Location** area at the bottom of the **Find: Autodesk Inventor Files** dialog box is where you can specify the folders to include in the search. Limiting the number of folders to search will accelerate the process. Knowing which folders to search may also avoid confusion. For example, suppose you are looking for a file named bracket.ipt for a drill press assembly. There may be a bracket.ipt file in other assemblies. Limiting the search to the drill press folder will prevent the other files from appearing in the list of found files. The folder and its entire path for the current search is displayed in the **Current Path File:** box. You may restrict the search to the current folder or include all subfolders by selecting the appropriate radio button: **Current Folder**, **SubFolders**, or **Related Files**. The **Look in:** drop-down list and the **Browse** button next to it are used to select drives and folders for searching.

Once you specify all search options, choose the **Find Now** button, located at the bottom of the **Find: Inventor Files** dialog box, to begin the file search. After you locate the existing Inventor file that you would like to open, double-click on the file name in the list box, or pick the file in the list box followed by selecting the **Open** button.

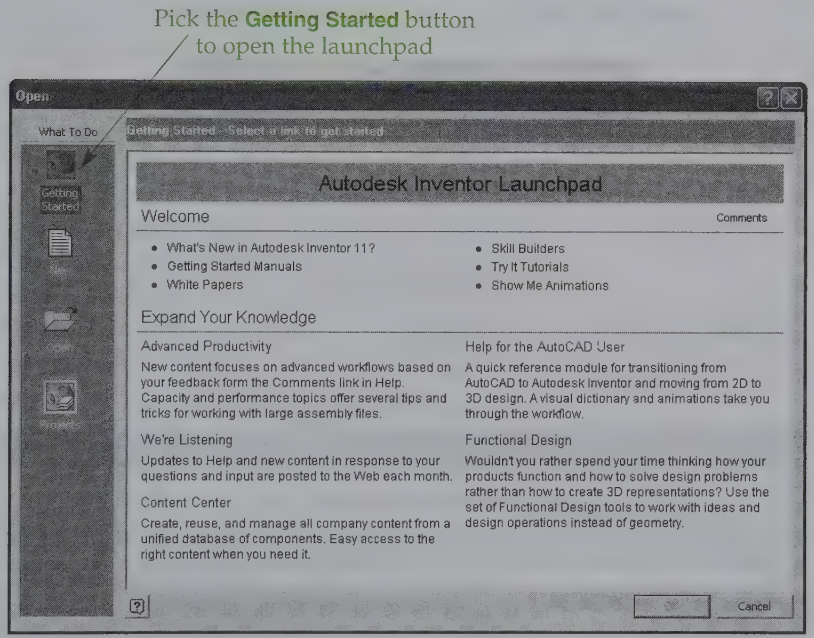


Complete the Exercise on the Student CD.

Getting Help

There are a variety of ways to get help with Inventor, including the **Autodesk Inventor Launchpad** of the **Open** dialog box, the **Help** pull-down menu and **Help** buttons. The **Autodesk Inventor Launchpad** of the **Open** dialog box, shown in Figure 1-38, is a major resource used to access Inventor help. To display the **Inventor Launchpad**, pick **Getting Started...** from the **File** pull-down menu, or if the open dialog box is already shown, select the **Getting Started** choice of the **What To Do** menu. The **Autodesk Inventor Launchpad** is an educational resource that allows you to pick from several learning tools. The **Welcome** section contains a number of options that open user manuals, tutorials, and animations. Some of these tools, such as **Skill Builders**, are available online and you must be connected to the Internet for them to function. The **Autodesk Inventor Launchpad** also has selections listed under **Expand Your Knowledge** that veteran Inventor users may find informative.

Figure 1-38.
The **Autodesk Inventor Launchpad** contains several different tools to help you understand various areas of Inventor.



NOTE

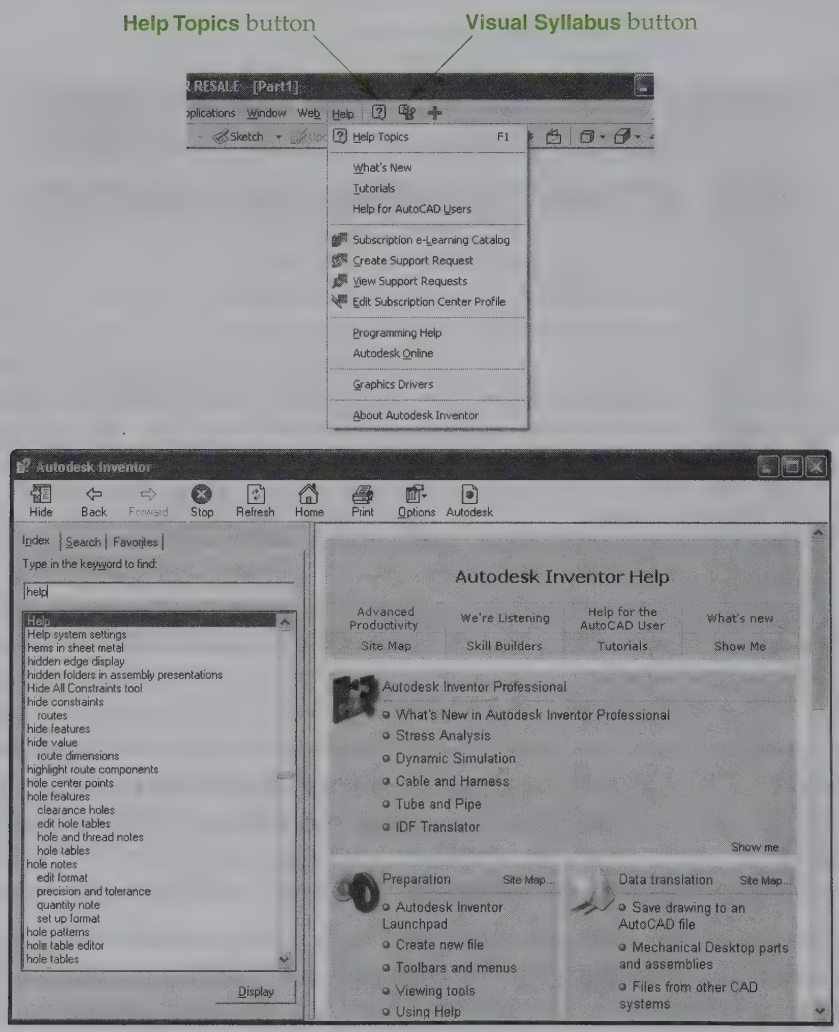
The **Autodesk Inventor Launchpad** contains several tutorials, documents, and manuals that are also accessible from the **Help** pull-down menu.

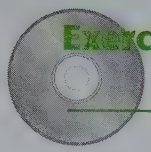
There are a variety of other ways to get help with Inventor, including the **Help** pull-down menu and **Help** buttons. The **Help** pull-down menu provides you with options that help you better understand Inventor. You can select from a variety of help

tools including links that open the Inventor help files, Inventor and Vault tutorials, the **What's New in Autodesk Inventor 11** dialog box, and other educational resources. You can also access Inventor help resources by picking one of the two support buttons next to **Help** on the pull-down menu bar. The **Help** button opens the Inventor help files, while the **Visual Syllabus** button opens the **Visual Syllabus** dialog box. Here you can select a specific topic, such as "creating an extruded feature", and the Inventor help files display a **Show Me** animation on how to create an extruded feature.

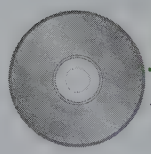
Picking a **Help** button, such as the **Help** button located near the lower left corner of the **Open** dialog box, opens the home page of the **Autodesk Inventor** help dialog box, shown in Figure 1-39. This feature of the program is similar to other Windows based help files, and allows you to further explore areas of Inventor and receive answers to any questions you may encounter. The **Autodesk Inventor** help dialog box is also accessed by selecting **Help Topics** from the **Help** pull-down menu; selecting the **Help Topics** button on the pull-down menu bar; pressing the [F1] key; or picking the **Help** button on numerous dialog boxes, the panel bar, or the browser.

Figure 1-39.
Practice using the **Help** pull-down menu, **Help Topics** button, and the **Visual Syllabus** button so you will know where to find specific information.





Complete the Exercise on the Student CD.



Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. Briefly discuss how part files are used.
2. Explain how assembly files are used.
3. Discuss how presentation files are used.
4. Briefly explain the function of drawing file templates.
5. Discuss the concept of parametric solid modeling.
6. What does it mean when the parametric concept is referred to as intelligence?
7. Describe *sketches.*
8. Define sketched features and give an example.
9. What is the initial sketched feature called?
10. Explain and give an example of placed features.
11. How are work features used?
12. Name and define three types of work features.
13. Describe and give an example of catalog features.
14. Briefly describe a derived component.
15. Explain and give an example of pattern features.
16. Describe a *rectangular pattern.*
17. Briefly describe a circular pattern.
18. What is a mirrored feature?
19. Why are constraints an important principle in Inventor and in parametric modeling?
20. Give an example of a constraint.
21. What does it mean when Inventor tells you if a sketch is over-constrained?
22. Describe a modeling failure.
23. What is it called and what does it mean when a model can also not contain enough constraints?
24. Describe fully constrained.
25. What are parameter-driven assemblies?
26. Describe adaptive parts in assemblies.
27. Give the function of a *help string.*
28. What are menu access keys and how do they work?
29. Give another name for a cascading submenu and explain how it works.
30. Identify the basic function of a dialog box.
31. How do you know when a pull-down menu item or a button opens a dialog box?
32. Give two other names for pop-up menus.
33. How do you access pop-up menus, and what does it mean when pop-up menus are said to be context sensitive?
34. Give the function of a panel bar and an example of its use.
35. How do you resize a panel bar?

36. By default the panel bar is docked in the upper-left corner of the work environment. How do you undock and redock the panel bar?
37. How is a grab bar represented?
38. What can you find in the browser bar?
39. Briefly describe *toolbars.*
40. Name the default toolbar found the first time you start Inventor, and what are the thin lines found in toolbars?
41. What is a *tooltip?*
42. What is a help string and when is it seen?
43. How do you know when a flyout is available, and what is its function?
44. Give the basic function of shortcut keys and how they are used, and what are the shortcut keys often called that require you to press the [Ctrl] key and an additional key, such as [Ctrl] and [O]?
45. What are hot keys and when are they used?
46. Very briefly define template and give another name for template.
47. How often should you save your work, and why is it important?
48. Name the dialog box where you save a new drawing that has never been saved before.
49. Name the primary tool used to access existing Inventor files.
50. Name the primary tool used to assess Inventor help and identify where it is found.

Problems

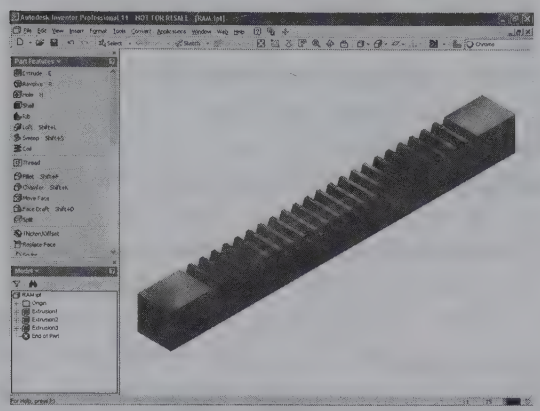
1. Write a short description of each of the four Inventor file types. Include information on file extensions and when each file type is used. Reflect on the capabilities of Inventor. How is Inventor similar and different from other CADD programs you may have used? Submit a typed hardcopy of your description to your drafting instructor or supervisor.
2. Write a brief report on each part model element and the importance of constraints and parametric associations. Reflect on your understanding of parametric modeling. Have you worked in parametric situations before? Submit a typed hardcopy of your report to your drafting instructor or supervisor.
3. Write a brief description of the Inventor Interface. What is an interface? What are the primary Inventor interface items, and how do they function? Reflect on the concept of a computer software interface. Are some of the Inventor interface items similar to those in other programs you have used? Submit a typed hardcopy of your description to your drafting instructor or supervisor.
4. Draw a freehand sketch of the standard Inventor screen display. Label each of the screen areas, and the interface items described in this chapter.
5. Launch Inventor and perform the following tasks:
   A. Begin a new Inventor part file, and explore the default part interface.
   B. Close the part file without saving.
   C. Begin a new Inventor assembly file, and explore the default assembly interface.
   D. Close the assembly file without saving.
   E. Begin a new Inventor drawing file, and explore the default drawing interface.
   F. Close the drawing file without saving.
   G. Begin a new Inventor presentation file, and explore the default presentation interface.
   H. Close the presentation file without saving.
   I. Exit Inventor.

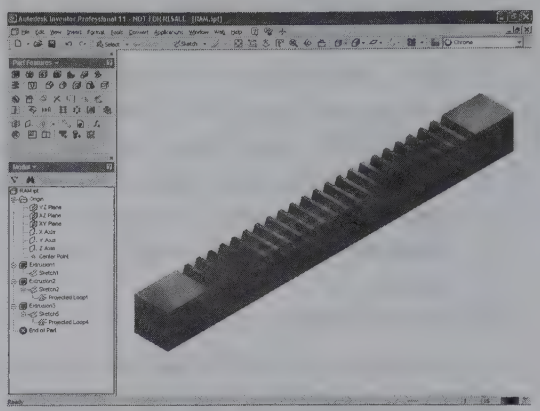
6. Launch Inventor and perform the following tasks:
   A. Begin a new Inventor file of your choice.
   B. Explore the Inventor pull-down menu system by picking any of the menu titles, and moving your cursor down and up through the menu options.
   C. Pick the **File** pull-down menu, and highlight the **New...** option. Notice the help string displayed in the status bar.
   D. Use a menu access key to access the **Open...** option of the **File** pull-down menu, and open the **Open File** window of the **Open** dialog box.
   E. Pick the **Cancel** button to close the **Open** dialog box.
   F. Explore cascading submenus by highlighting the **Toolbar** option of the **View** pull-down menu.
   G. Press the [Esc] key, or pick outside of the menu to close the **View** pull-down menu.
   H. Exit Inventor.

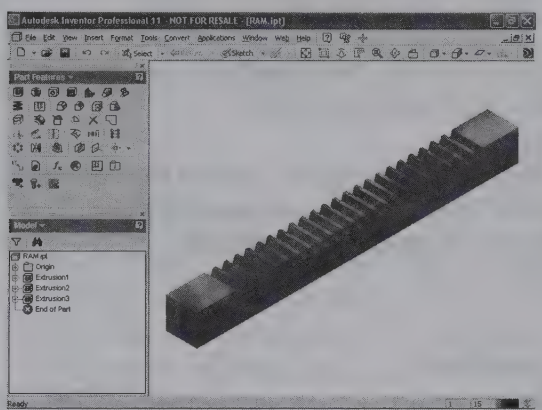
7. Launch Inventor and perform the following tasks:
   A. Open blade_main.ipt from the following directory: Autodesk/Inventor 11/Samples/Model/Assemblies/Scissor/Scissor Components/blade_main.ipt.
   B. Locate and explore the panel bar.
   C. Pick the panel bar title, **Part Features**, and select the **Expert** option from the **Shortcut** menu. Observe the changes made to the panel bar.
   D. Locate and explore the default browser bar.
   E. Drag the **End of Part** indicator up and above Sketch4, and notice the changes.
   F. Pick the **Expand** button to the left of the **Origin** folder to expand all children.
   G. Right-click on the **Origin** folder, and select **Collapse All Children**.
   H. Close blade_main.ipt without saving.
   I. Exit Inventor.

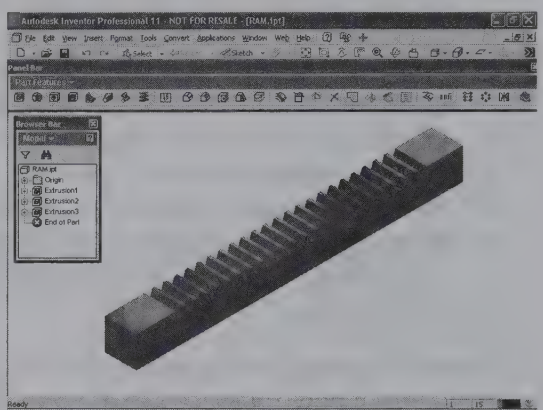
8. Launch Inventor and perform the following tasks:
   A. Begin a new Inventor file of your choice.
   B. Locate and explore the **Inventor Standard** toolbar.
   C. Using one of the techniques described, open the **Inventor Collaboration** toolbar, and dock it by double-clicking the toolbar name.
   D. Place your cursor over several toolbar buttons and observe the tooltip and help strings provided.
   E. Pick one of the flyout buttons located in the **Inventor Standard** toolbar, pick one of the flyout options to initiate a tool, and modify the flyout button displayed.
   F. Exit Inventor.

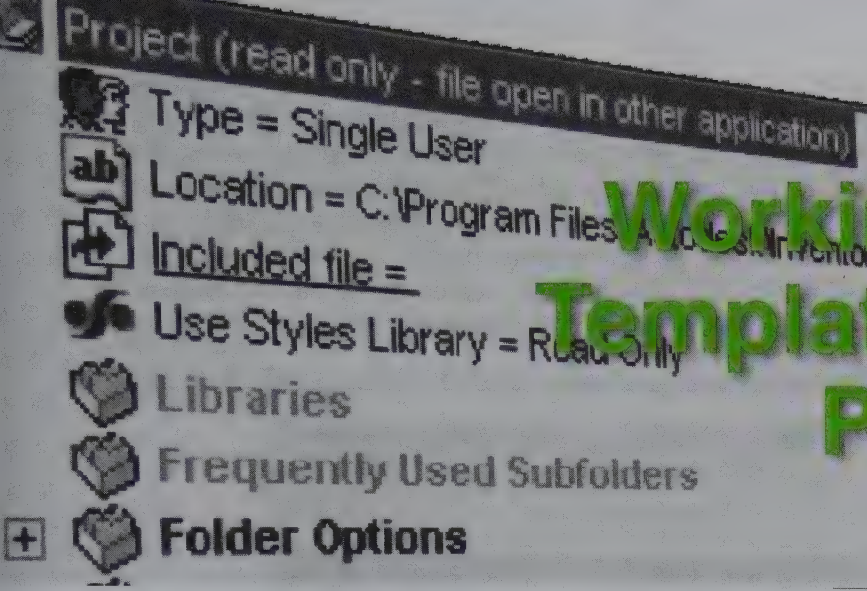
You should try to use the panel bar in Expert mode as much as possible. This will allow you to pick the icons faster, since you will not have to use the scroll bar. Also, this method will provide more room for the graphics window. Other factors that affect the size of the graphics window are monitor size, screen resolution, and docking the panel and browser bars.











CHAPTER 2

Working with Templates and Projects

Learning Objectives

After completing this chapter, you will be able to do the following:

- Access and create template files.
- Use the Project Editor and create projects.

Inventor templates and projects are very important design preparation tools that help you effectively manage files and file characteristics, quickly access particular work environments, and operate in a shared-design setting. You will become familiar with design settings that are specific to part, assembly, presentation, and drawing files as you progress through different design stages. However, before you begin working on an Inventor file, you should understand and have the ability to work with templates and projects.

Templates

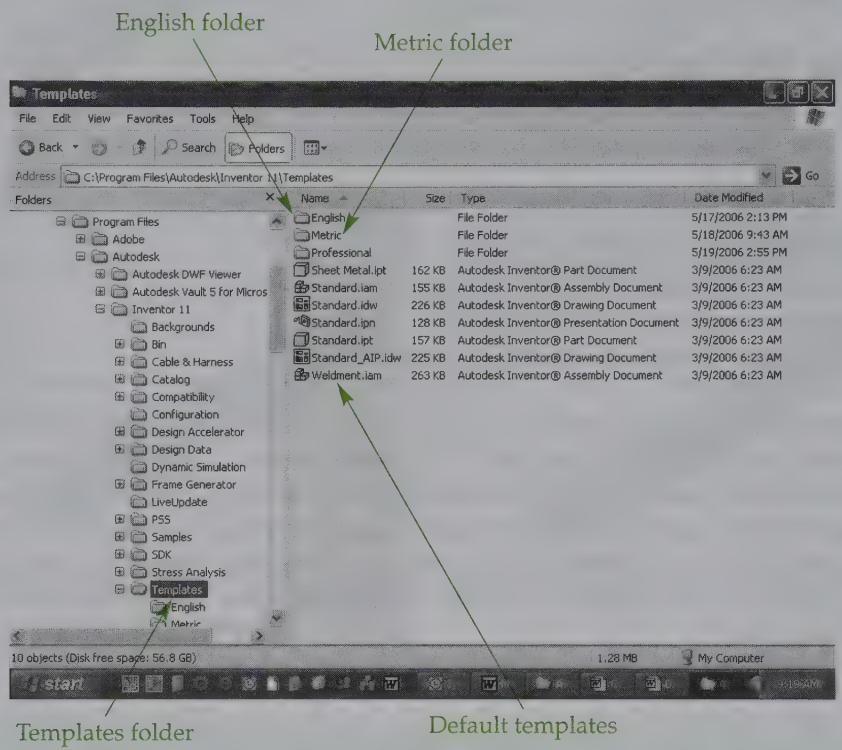
Template options and specifications vary depending on the file type and drafting standards. For example, you might have a template for designing steel parts and another for aluminum parts, or a template for metric drawings and another for U.S. Customary (inch) drawings. Chapter 1 describes how to access template files using the **New File** window of the **Open** dialog box. The various templates provided by Inventor are adequate for many applications; however, to increase efficiency and maintain uniformity you should create your own templates based on specific industry, school, and company standards.

Developing your own template can save time and effort, because the original work environment settings you establish are already set every time you open the template. You can develop a template for each of the four Inventor file types, including subfile templates for sheet metal parts and weldments. For example, your part file templates can include unit settings and various work environment options such as grid spacing and styles, and your drawing file templates can include unit settings, a title block, and a border. The process of creating a template first involves opening an existing template file. The next step is to adjust the work environment and template settings.

All the tabs and templates found in the **New File** window of the **Open** dialog box are stored in folders in the path Autodesk/Inventor 11/Templates, and can be accessed

using Microsoft Windows Explorer, as shown in Figure 2-1. The folders correspond to the tabs shown in the **New File** window of the **Open** dialog box. The **Default** tab does not have a separate folder. The part, assembly, presentation, and drawing file templates, found under the **Default** tab, are located in the **Templates** folder, in the path Autodesk/Inventor 11/Templates. In contrast to the **Default** tab, the **English** and **Metric** tabs each correspond to separate folders. The part, assembly, presentation, and drawing file templates are found in the English and Metric folders, located in the paths Autodesk/ Inventor 11/Templates/English and Autodesk/ Inventor 11/Templates/Metric.

Figure 2-1.
Use Windows Explorer to find the folder that contains the templates for Inventor.



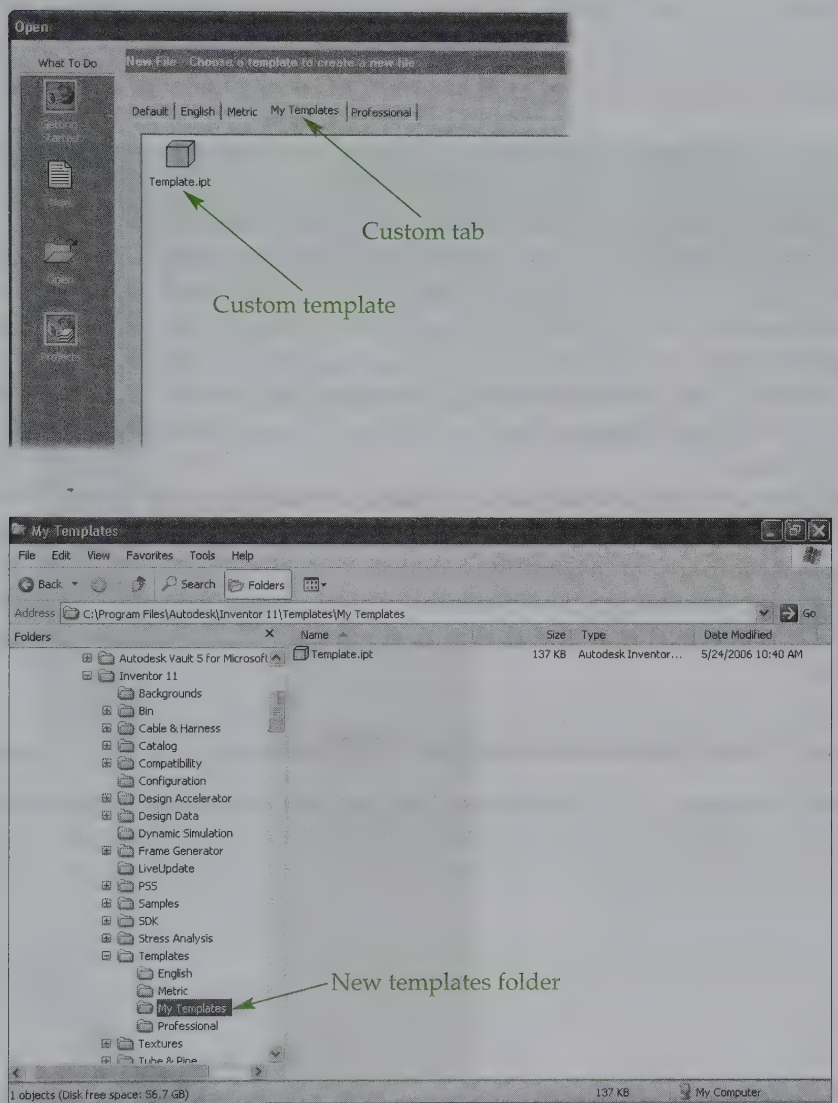
Creating New Templates

In most cases, templates are created using preexisting Inventor part, assembly, presentation, or drawing file templates. For example, you may want to begin with Standard.ipt for a part or Standard.iam for an assembly. There are a few options for creating templates and tabs, such as using Microsoft Windows Explorer to copy and rename folders and existing template files. The following describes one of the most effective techniques:

1. Open an available template file from one of the tabs located in the **New File** window.
2. Select **Save** from the **File** pull-down menu to display the **Save As** dialog box.
3. Use the **Save As** dialog box to locate the **Templates** folder in the designated file path, most often, Autodesk/Inventor 11/Templates. Navigating to the **Templates** location is accomplished by opening drives and folders in the **Save in:** drop-down list.
4. Once you find the location of the **Templates** folder, select the **Create New Folder** button and type a name over the existing **New Folder** default name.
5. Double-click on the folder you just created, or press the [Enter] key twice.
6. Specify a template file name in the **File name:** edit box, and select the **Save** button. Now, when you launch Inventor and access the **New File** window of the **Open**

dialog box, the template tab and files you created are visible. You also see the tab as a folder under Autodesk/Inventor 11/Templates containing the template file you have created. For example, Figure 2-2 shows a custom tab named **My Templates**, with a custom template named Template.ipt in the **New File** window of the **Open** dialog box, and the corresponding folder and file in Microsoft Windows Explorer.

Figure 2-2.
The tab **My Templates** and the corresponding folder were created to store custom templates.



NOTE

A **New File** window tab only displays if a template file is present in the template folder.

PROFESSIONAL TIP

To access custom templates using the **New** flyout button, modify or replace the default Standard .ipt, .iam, .idw, and .ipn files located in the **Templates** folder.

EXERCISE 2-1

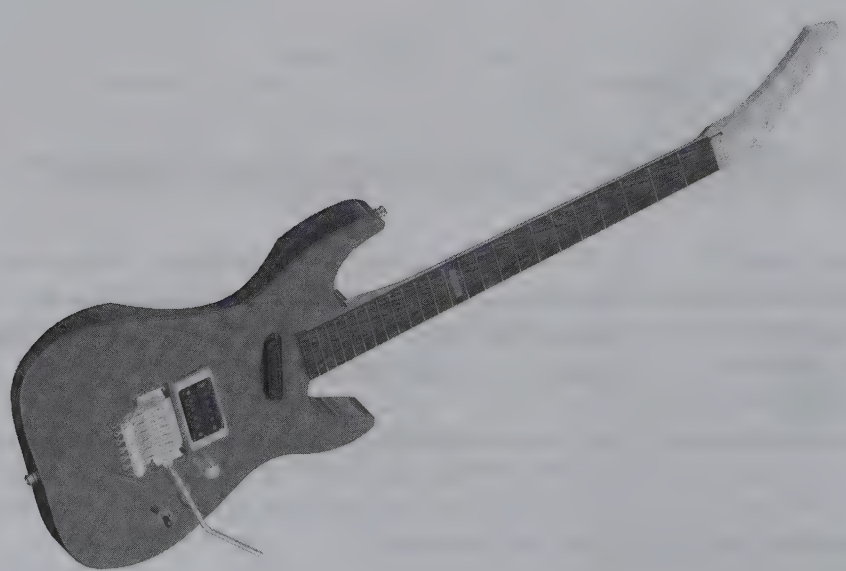
Complete the Exercise on the Student CD.

Projects

The parametric nature of Inventor requires that documents are associated with each other and that information can be referenced from one design stage and file to another. For example, an assembly references component files, a presentation references an assembly file, and a drawing references a part file. Inventor uses projects to effectively reference and deal with these parametric file associations. ***Projects*** are used to manage and organize Inventor files, and allow Inventor to find and provide connections between files.

Inventor projects are, as the name implies, virtual design projects. For example, the guitar shown in Figure 2-3 is a design project that consists of numerous parts, a couple of subassemblies, an assembly, a couple of technical illustrations, and a complete set of working drawings. The Inventor project created specifically for this design contains the same items, but in the electronic form of part, assembly, presentation, and drawing files.

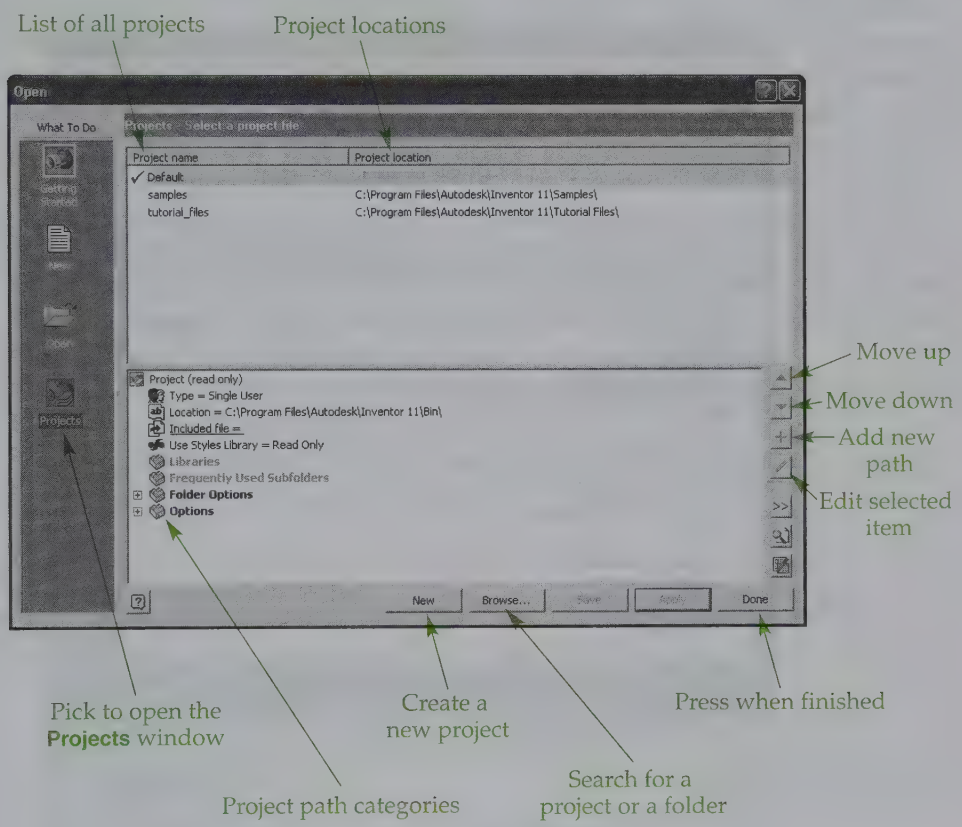
Figure 2-3.
Using a project to organize all of the folders, files, and relationships between files necessary to build a part like this will help you become an efficient and successful Inventor user. (Model courtesy of Ethan Collins)



In most situations you should create a project for each design. Separate projects allow you to better organize potentially hundreds of files. You can create and use as many projects as necessary to manage your work in an efficient manner. Projects are created, edited, and activated using the **Project Editor**.

The **Project Editor** is available in the **Open** dialog box, as shown in Figure 2-4, by selecting the **Projects** option of the **File** pull-down menu, or if the open dialog box is already shown, select **Projects** from the **What To Do** menu. You can also access the **Project Editor** by selecting the Microsoft Windows **Start** button>**Programs**>**Autodesk** >**Autodesk Inventor 11**>**Tools**>**Project Editor** or by right-clicking on a project file and selecting **Edit**. Project files carry the extension .ipj. See Figure 2-5.

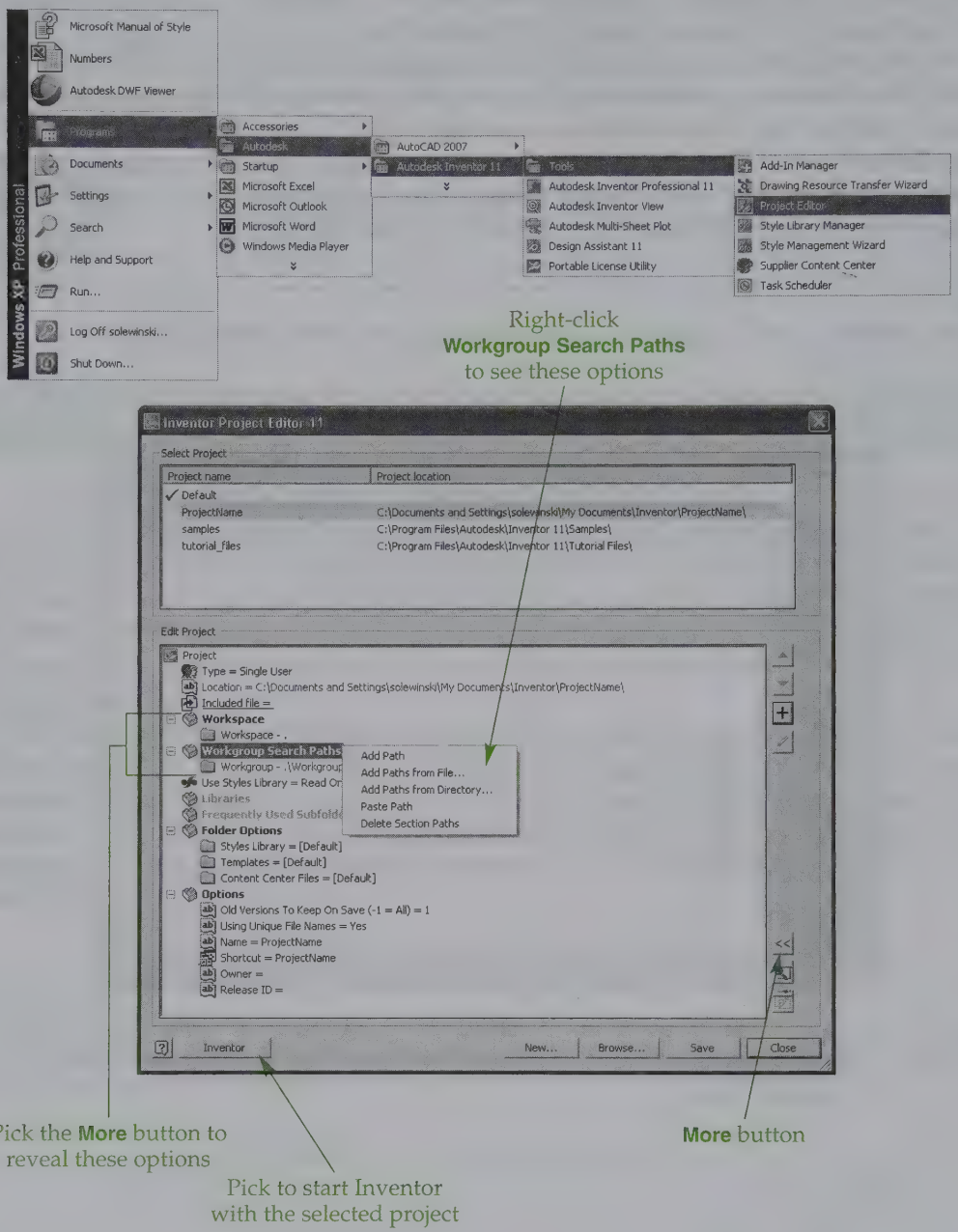
Figure 2-4.
This project editor is available by picking the **Projects** icon in the **Open** dialog box.



NOTE

Opening the **Project Editor** from outside of the **Open** dialog box does not open the **Open** dialog box. Instead, it launches only the **Inventor Project Editor 11** dialog box. The only difference between the **Project Editor** found in the **Open** dialog box and the **Inventor Project Editor 11** dialog box is the **Inventor** and **Close** buttons, located below the lower list box, and the lack of **Apply** and **Cancel** buttons. Choosing the **Inventor** button launches Inventor with the selected project active.

Figure 2-5.
The Inventor **project editor** may be started alone by picking the menu selection as shown.



The Project Editor

The Inventor **Project Editor** contains two list boxes, or window panes, and several buttons. The upper list box in the **Project Editor** holds two lists. **Project name** displays a list of available projects according to project names, and **Project location** provides the path, or location, of each project. Clicking once on the name of a project allows you to view the project specifications in the lower list box. Double-clicking the project name or selecting the project and picking the **Apply** button activates the project. When a project is active, a check mark is visible to the left of the project name. Initially, three Inventor project files are available: **Default**, **samples**, and **tutorial_files**. The **Default** project is initially active when you open the **Project Editor** for the first time.

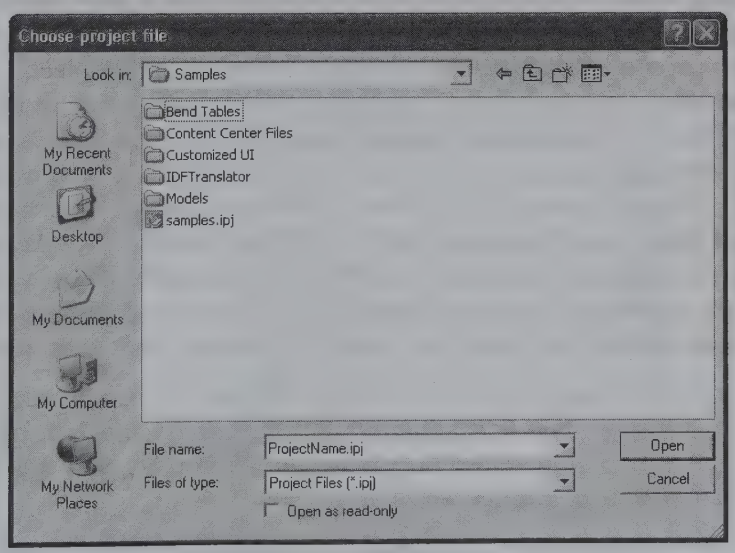
NOTE

The project that is active when the **Project Editor** is closed is the active project until you activate a different project. An error message tells you if files are not located in the active project. You must close all Inventor files before you can activate a different project.

When you right-click on an inactive project in the upper list box, the **Rename**, **Browse...**, **New...**, and **Delete** options are available, except for **Default**, **samples**, and **tutorial_files**. The function of each option is explained as follows:

- **Rename**. Select this option to highlight and change the project name. Type a new name over the existing name, and press [Enter].
- **Browse...** Picking this option opens the **Choose project file** dialog box. See Figure 2-6. You can also open the **Choose project file** dialog box by selecting the **Browse** button below the lower list box of the **Project Editor**. The **Choose project file** dialog box allows you to select a project that is not currently shown in the upper list box of the **Project Editor**, but is still available somewhere on your computer or the network. The **Choose project file** dialog box only allows you to access Inventor project files, identified with the extension .ipj, and contains an **Open as read-only** check box. If selected, the **Open as read-only** check box opens the project file as *read-only*; which allows you to view, but not change the file. After finding the project, double-click on the project name, or select the file and pick **Open**. Accomplishing this task allows Inventor to create a shortcut to the file in the **Projects** folder, and lists the project in the upper list box.
- **New**. Selecting this button opens the **Inventor project wizard** dialog box, which is used to create a new project file.
- **Delete**. Selecting this button removes the specified project from the list, but the project can be retrieved using the **Browse...** option.

Figure 2-6.
The **Choose project file** dialog box contains the typical file-locating list boxes and folders.



NOTE



You cannot rename or delete the active project. Only editing tools that apply to the currently selected project are available while other tools are shaded.

The lower list box of the **Project Editor** is composed of a **Project** folder containing project shortcuts, specifications, and options. In order to view all possible project data, you must pick the **More** button located to the right of the list box. The items in the lower list box of the **Project Editor** can be confusing. Just remember that projects are used to organize and link files together, and the purpose of the following items is to ensure that files can be found. You can use as few or as many of these options depending on the location of files:

- **Type** displays the kind of project in reference to the file access characteristics. The file type identifies where files are located and who has access to files. By default, only the **Single User** type is applicable, because encourages you to use Autodesk Vault to manage design team projects. A **Single User** type is a project where all files, except for any possible library files, are kept in a single location. If you have a need to access project types related to earlier Inventor versions, or if you do not have Vault loaded, turn on the **Enable creation of legacy project types** check box. You can access this by picking **Tools>Application Options>General** tab. After placing a checkmark in the **Enable creation of legacy project types** check box, you have access to the following types:
  - **Vault**. Select this mode to create a relationship between the active project and Vault (a separate file management application). You must install Vault in order to use the application with Inventor and Inventor projects. For more information regarding Vault, refer to the Autodesk Web site and Vault help files.
  - **Semi-isolated**. Select this mode to create a project with both shared workgroup and personal workspace areas. A semi-isolated multiuser setting establishes a master project location that all design team members can access, and personal workspaces for each individual.
  - **Shared**. Select this mode to specify a shared project in which all design team members can access and change files.
- **Location** provides the path, or location, of the selected project.
- **Included file** allows one additional project file and its paths to be included in a given project by combining the information in the included file and the project file together. You can identify an included file any time you want the selected project to have access to the files in another project. Right-clicking on an **Included File** reveals the **Open**, **Edit**, and **Delete** options described as follows:
  - **Edit**. Use this option to create an included file path. When you pick the **Edit** option, a **Browse** button is displayed. Picking the **Browse** button opens the **Choose project file** dialog box used to locate and select an included file.

NOTE



The **Edit selected item** button, located to the right of the lower list box, also allows you to edit the **Included file** path location.

  - **Open**. Pick this option to display the included file path in the **Project Editor**.
  - **Delete**. Choose this option to delete the included file path.

- **Workspace** is displayed when you pick the **More** button. The *Workspace* is the default directory where Inventor files are located in the project. If all design project files are saved to one place, as they typically should be, a **Workspace** is the only working directory in a given project. However, the **Workspace** may be one of many directories if files are located in several areas. In addition, the **Workspace** is your exclusive area to operate if you are working within a design team. Though rarely needed, you can modify the workspace by right-clicking on and selecting the appropriate pop-up menu option, or by picking the desired button located to the right of the lower list box.

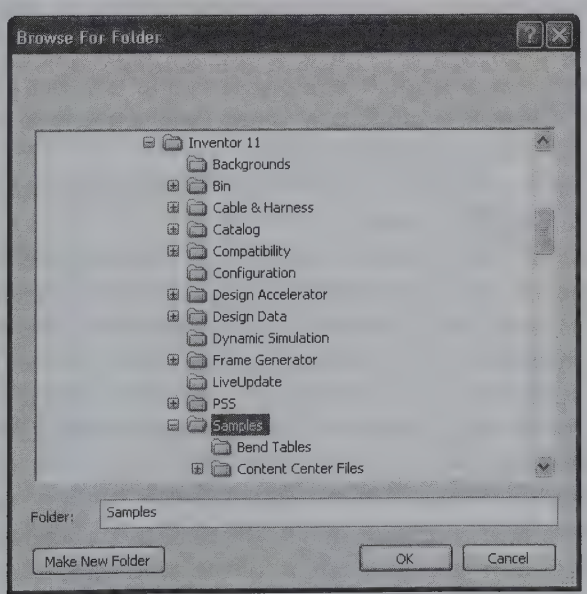
NOTE



A project can contain only one **Workspace**.

- **Workgroup Search Paths** is displayed when you pick the **More** button. *Workgroup Search Paths* allow projects to locate files on a server or other computers on a network. The files are found in the shared environment of the network and are used primarily in a design team setting. Right-clicking on **Workgroup Search Paths** provides you with the following options (See Figure 2-5):
  - **Add Path**. Pick this option to add a path to the specified selection. After choosing the **Add Path** option, you can change the default name and find a folder using the **Browse** button located to the right of the path. The **Browse for Folder** dialog box, shown in Figure 2-7, allows you to add a path to the specified selection from an existing file.
  - **Add Paths from File...** Selecting this option opens the **Choose project file** dialog box used to locate and select a project file to add to the workgroup.
  - **Add Paths from Directory...** Choosing this option opens the **Browse For Folder** dialog box, as shown in Figure 2-7. This box allows you to locate a folder from anywhere on your computer or the network.

Figure 2-7.
Use the **Browse For Folder** dialog box to select a workgroup search path.



- **Paste Path**. Picking this option allows you to paste a path from one location to another. To use this tool, right-click on the path that you want to paste to the new location. Next, select either **Cut** or **Copy** from the menu items. Finally, locate the area where you want to paste the path, right-click and select **Paste Path**.
- **Delete Section Paths**. Selecting this choice deletes every path located in the workgroup.

Right-clicking on a path located under **Workgroup Search Paths** reveals the following options:

- **Edit**. This option allows you to edit the path name and location. Selecting **Edit** also displays the **Browse For Folder** button, located to the right of the edit box.

NOTE



The **Add new path** button, located to the right of the lower list box, also allows you to add a new path to the specified selection. The **Edit selected item** button, located to the right of the lower list box, also allows you to edit the path name and location.

- **Add Path**. Pick this option to add a path to the specified selection. After choosing **Add Path**, you can change the default name and find a folder using the **Browse** button located to the right of the path. The **Browse** button opens the **Browse For Folder** dialog box.
- **Delete**. Pick this choice to delete the specified path from the directory.
- **Move Up**. Selecting this option moves the selected folder up in the list of folders, and can also be done by picking the **Move Up** button to the right of the list box.
- **Move Down**. Choosing this option moves the selected folder down in the list of folders. You can also accomplish this task by picking the **Move Down** button to the right of the list box.

- **Use Style Library** describes how styles are used when working with files in the selected project. The following options are available when you right-click on **Use Style Library**:
  - **Yes**. If selected, style libraries are used and can be modified in the files found in the given project.
  - **Read Only**. When you pick this option, style libraries can be used, but not changed. The CAD administrator typically sets the style libraries to **Read Only** to maintain a standard set of styles.
  - **No**. Pick this option if you do not want to use style libraries when working with files in the given project.
- **Libraries** allow Inventor to locate library files that are used in a project or several different projects, but are not modified as part of the project design process. The elements of a library are generally shared and may include standard, often-used parts, components from other programs, such as Mechanical Desktop, or any other libraries that you may need to reference. Right-clicking on **Libraries** reveals the same options available with **Workgroup Search Paths**, except for the **Add Paths from Directory...** option. Right-clicking on one of the folders located under **Libraries** gives these menu options, in addition to **Edit** and **Add Path**:
  - **Add Proxy Path...** Choosing this option opens the **Browse For Folder** dialog box, which allows you to locate a folder from anywhere on your computer or the network.
  - **Cut**. Use this option to remove the specified path. The path can then be pasted in a new location.

- **Copy**. Select this option to copy the specified path for use in a new location, without removing the path from the initial directory.
- **Paste**. This option allows you to paste a file from one location to another.
- **Delete**. Pick this choice to delete the specified path from the directory.

- **Frequently Used Subfolders** contain folders that are specific to the selected project, and contain files that need to be accessed often. The same options available when you right-click on **Workgroup Search Paths** are found when you right-click on **Frequently Used Subfolders**.
- **Folder Options** contains the **Style Library**, **Templates**, and **Content Center Files** folders. The folders allow you to customize access to styles, templates and content center files. For example, if you only want specific templates to be available when the selected project is active, right-click on the **Templates** folder, pick the **Edit** pop-up menu option, and use the **Browse** button to locate the **Templates** folder you want to use. The process is the same with the **Style Library** and **Content Center Files** folders.
- **Options** establishes the default settings for the specified project. The options may include the following:
  - **Old Versions To Keep On Save**. When you save a file in the active project, an **OldVersions** folder is created. This folder contains backup versions of the file that are redefined each time the file is saved. When the specified maximum number of old versions is saved, the oldest version is removed to accommodate a newer version. To specify the maximum number of backup file versions to keep in the **OldVersions** folder, pick the **Old Versions To Keep On Save** option. Then, pick the option again, enter the maximum number of old file versions in the edit box, and select outside the edit box to make the change.
  - **Using Unique File Names**. Use this setting to define what happens if files with the same name are added to a project. If you do not want any duplicate file names to be used in the project, right-click on **Using Unique File Names** and pick the **Yes** pop-up menu option. Selecting **Yes** indicates that all files in the project will have unique names. If files in the project may have the same name, right-click on **Using Unique File Names** and pick the **No** pop-up menu option. If you choose **No** and files with the same name are found in the project, the **Resolve Files** dialog box is displayed allowing you to identify exactly which file you want to use.
  - **Name**. This section displays the project name.
  - **Shortcut**. This option displays the name given to the project shortcut. To edit the shortcut name, right-click on the existing shortcut name and select **Edit** or pick the shortcut name twice. To delete the specified shortcut name, right-click on the shortcut name and select **Delete**.
  - **Owner**. This setting identifies the owner of the project, such as the CAD administrator. If you want all members of the design team to know the name of the person managing the project, pick to highlight **Owner**, then pick it again and enter the desired name.
  - **Release ID**. Use this option if you need to further classify a project. A project release ID can help users identify the project they have accessed.
- **Find duplicate files**. Pick this button to access the **Non-Unique Project File Names** dialog box. The **Non-Unique Project File Names** dialog box is a reference source that displays all the files that have the same names in the project.
- **Edit Content Center Configuration**. Select this button to open the **Configure Libraries** dialog box. The **Configure Libraries** dialog box allows you to add, remove, import, and modify content center libraries associated with the selected project.



Complete the Exercise on the Student CD.

Creating New Projects

Projects are created within the **Project Editor**. You can access the **Project Editor** from the **Projects** window of the **Open** dialog box, by selecting the Microsoft Windows **Start** button>**Programs**>**Autodesk**>**Autodesk Inventor 11**>**Tools**>**Project Editor,** or by right-clicking on a project file and selecting **Edit**.

To create a project, right-click on a project name or a project location and select the **New...** option, or select the **New** button, located under the lower list box of the **Project Editor**. Both methods open the **Inventor project wizard**, shown in Figure 2-8, which allows you to create a new project file.

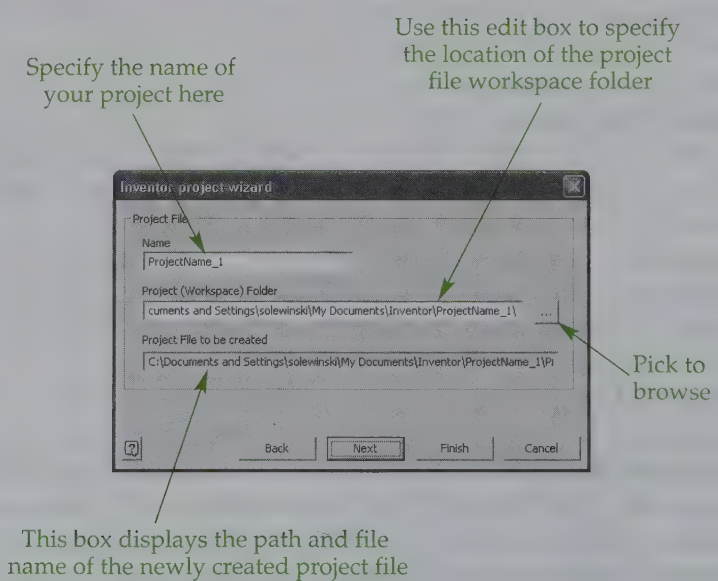
NOTE



By default, you can only create a single user project type. If you need to create other project types, related to earlier Inventor versions, or if you do not have Vault loaded, access the **Enable creation of legacy project types** check box from the following location: **Tools**>**Application Options**>**General** tab and select the **Enable creation of legacy project types** check box. Additional pages are displayed in the **Inventor project wizard**.

Assuming you do not select the **Enable creation of legacy project types** check box from the **General** tab of the **Application Options** dialog box, the first page of the **Inventor project wizard** contains the options shown in Figure 2-8.

Figure 2-8.
The **Inventor project wizard** dialog box is where you can assign a name and folder for a project.



NOTE

You have the option of canceling the **Inventor project wizard** at any time by picking the **Cancel** button. Also, picking the **Back** button allows you to return to the last Inventor project wizard page to modify or check selections.

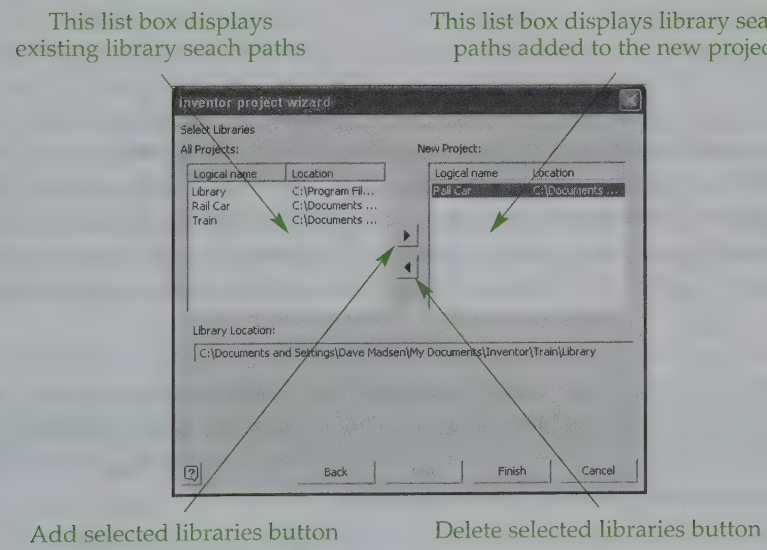
Once you specify all the options in the **Project File** page of the **Inventor project wizard,** pick the **Next** button to get the **Select Libraries** page of the **Inventor project wizard,** as shown in Figure 2-9.

This page allows you to add existing library search paths to the new project file. To add a library search path, select the libraries you would like to add from the **All Projects:** list box. Then pick the **Add selected libraries** button, located between the **All Projects:** and **New Project:** list boxes. To remove a library search path, select the libraries you would like to remove from the **New Project:** list box. Then pick the **Delete selected libraries** button, located under the **Add selected libraries** button.

After you specify all the new project options, pick the **Finish** button to complete the creation of the new project file. The project you just created is listed in the upper list box of the **Project Editor,** and is initially not active. Use the tools and options previously described to customize your new project.

EXERCISE 2-3 Complete the Exercise on the Student CD.

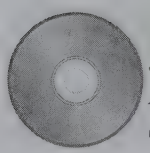
Figure 2-9.
The **Select Libraries** window of the **Inventor project wizard** dialog box is where projects are assigned to other projects.



NOTE



The project you have just created is used throughout this book. Store all of your exercise and problem files in the **Exercises and Problems** folder. You do not have to have this project to create models and drawings. However, if you place all of your exercise and problem files in the **Exercises and Problems** folder, searching for files should be minimal.



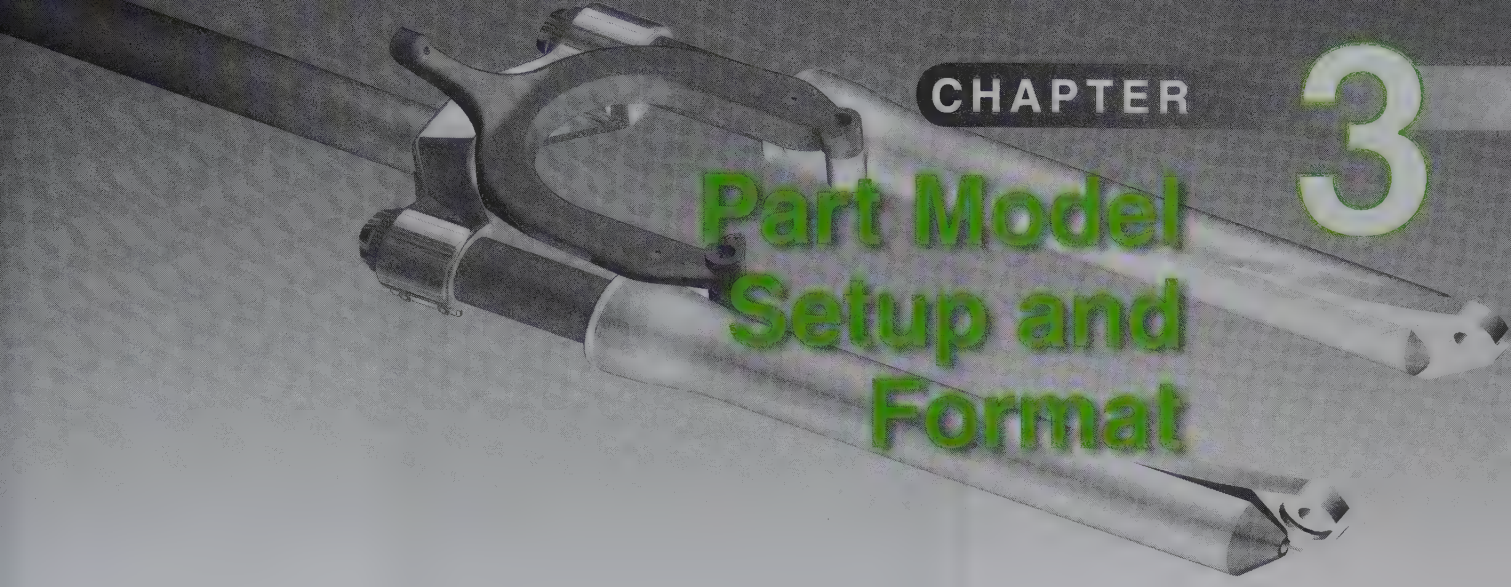
Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. Briefly discuss how developing your own template can save time and effort, and give an example of a template and its contents.
2. List the steps for creating templates and tabs using Microsoft Windows Explorer to copy and rename folders and existing template files.
3. Define *projects.*
4. Explain and give an example of what it means to say that Inventor projects are virtual design projects.
5. Give the function of the **Project name** list box in the Project Editor.
6. How do you view the project specifications in the Project Editor?
7. Give the function of the **Project location** list box in the Project Editor.
8. Give two ways to activate a project in the Project Editor?
9. Name the default directory where Inventor files are located in the project.
10. Identify where projects are created, and provide two options for finding this feature.

Problems

1. Add two custom **New File** window template tabs to the **Open** dialog box. Name one tab "INCH Templates" and name the other tab "METRIC Templates". These template tabs will be used to store exercise and problem templates, when you create them in future chapters. Remember that template tabs are not displayed in the **New File** window of the **Open** dialog box until you add template files to the template folders.
2. Create a new project named Problem 2-2. Place the project in a new folder named Problem 2-2, in a drive somewhere on your hard drive or the network. Set the **Use Styles Library** to **Read Only**. Include your Exercises and Problems project file. Delete the project you just created. Use the **Browse** function to locate the deleted project and return it to the **Project Editor** list.



CHAPTER 3

Part Model Setup and Format

Learning Objectives

After completing this chapter, you will be able to do the following:

- Work with part model iProperties.
- Use model view tools.
- Describe styles and work with model styles.
- Adjust part model document settings.
- Set part model application options.

Once you have a basic understanding of parametric solid modeling, the Inventor interface, and work environments, you are ready to begin the process of creating a model. Though you can begin a model using a default part template and the default part model work environment settings, it is a good practice to use a part template that contains predefined settings and options according to specific industry and company standards. Additionally, you should become familiar with many of the set-up, format, and work environment tools and options that are used in Inventor to develop part models.

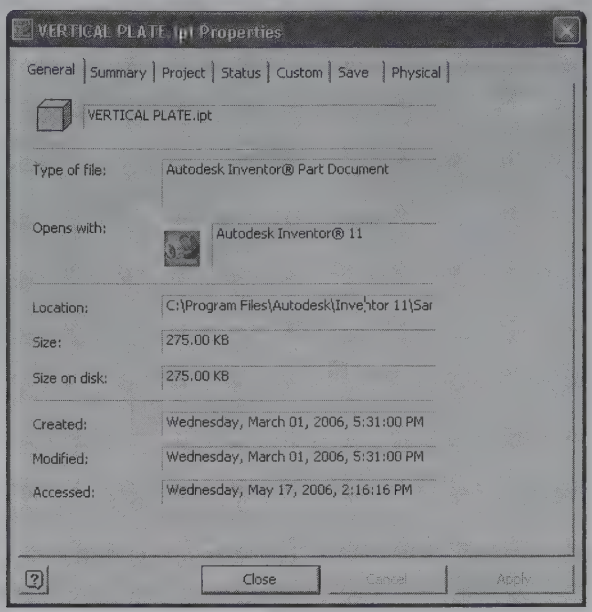
This chapter covers information you need in order to create part models. You will create templates with specific part model settings to use for exercises and projects. You will also explore a variety of part and model work environment tools that allow you to effectively generate Inventor models.

iProperties

Inventor ***iProperties*** are used to define a variety of file and design characteristics. iProperties represent a range of important file information from the file type, to the file name, and physical properties of the material used to manufacture the product. Picking the **iProperties...** option from the **File** pull-down menu opens the **Properties** dialog box for the specified Inventor file. See Figure 3-1. You can also open the **Properties** dialog box in Microsoft Windows Explorer or Inventor **Design Assistant** by right-clicking on any Inventor file and pick **iProperties** from the pop-up menu option.

Inside the **iProperties** dialog box are different tabs that represent sets of Inventor file properties. These properties are used throughout the design process to organize, manage, and document specific model property data. They are also used for developing title blocks, parts lists, and bills of materials.

Figure 3-1.
The items in the **General** tab of the **Properties** dialog box cannot be modified.



PROFESSIONAL TIP



You can specify properties at any time during the design process. However, it is most effective to define properties when you begin to develop a model. Additionally, specify many common properties used throughout a project in a template file, then every time you create a new part, the properties are the same.

The following describes each of the tabs located in the **Properties** dialog box:

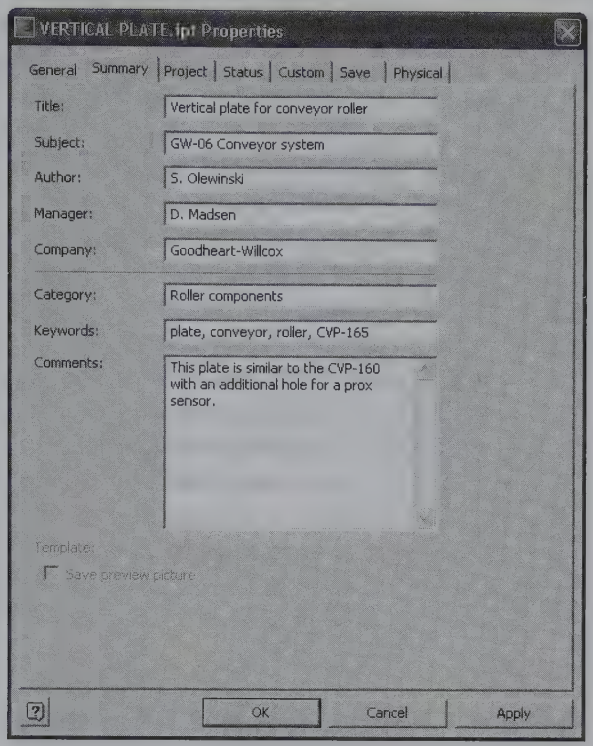
- **General.** This tab, shown in Figure 3-1, provides general information regarding the selected Inventor file. Here you can quickly review the file name, type, location, size, and other data, but you cannot modify any of these specifications using the **Properties** dialog box.
- **Summary.** This tab, shown in Figure 3-2, contains a number of edit boxes that allow you to define summary properties for the specified file. Summary properties are used to organize and help control specific file information such as the title, author, and company name. In addition, summary properties help you develop and manipulate title blocks, parts lists, reports, and bills of materials. To use the properties in the **Summary** tab, pick inside the edit boxes and type your information.

NOTE



The author is automatically specified as the name provided when Inventor was loaded, or as the name of the individual who initially created the file if you are working in a networked environment. Still, the default name can be modified.

Figure 3-2.
Add information in the **Summary** tab for efficient file management.



- **Project.** This tab contains a number of edit boxes and a creation date drop-down list that allow you to define project properties for the specified file. See Figure 3-3. Project properties are used to organize and help control additional file information such as the part number and engineer's name. Project properties also help you develop and manipulate title blocks, parts lists, reports, and bills of materials. To use the **Project** tab, pick inside the edit boxes and type the properties. The **Creation Date:** drop-down list can be used like an edit box, or you can pick the arrow and select a date from the pop-up calendar.

NOTE

The file name is automatically assigned as the part number if you do not define another number for the part. A real number must be used for an estimated cost.

- **Status.** The tab shown in Figure 3-4 contains a number of edit boxes and drop-down lists that allow you to define the specified file status properties. Status properties are used to organize and help control information related to design standing. Data such as the design state, checks, approvals, and file status are available. Status properties also help you develop and manipulate title blocks, parts lists, reports, and bills of material. To use the properties in the **Status** tab, pick inside the edit boxes and type your information, select an option from the drop-down list, or pick the date arrow and select a date from the pop-up calendar.

NOTE

The **File Status** area of the **Status** tab provides information about the file condition for use in a collaborative work environment and is only available with multiuser projects.

Figure 3-3.
Information in the **Project** tab of the **Properties** dialog box describes the specified file.

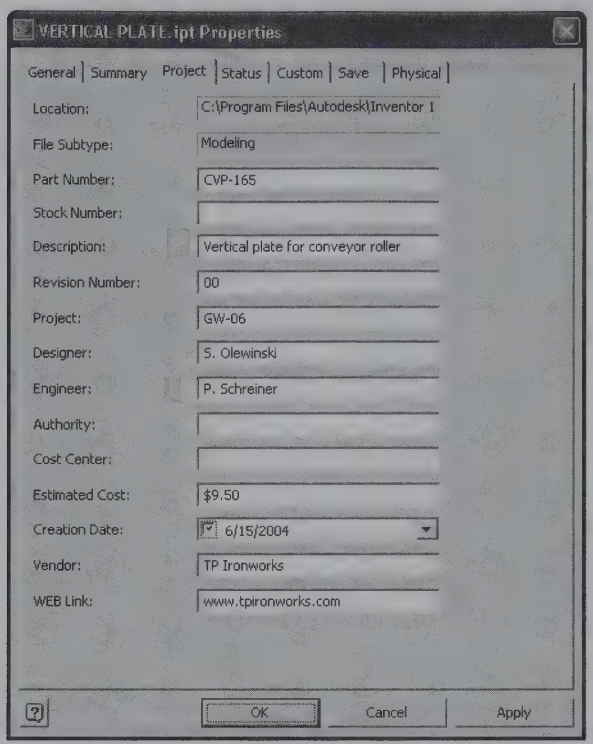
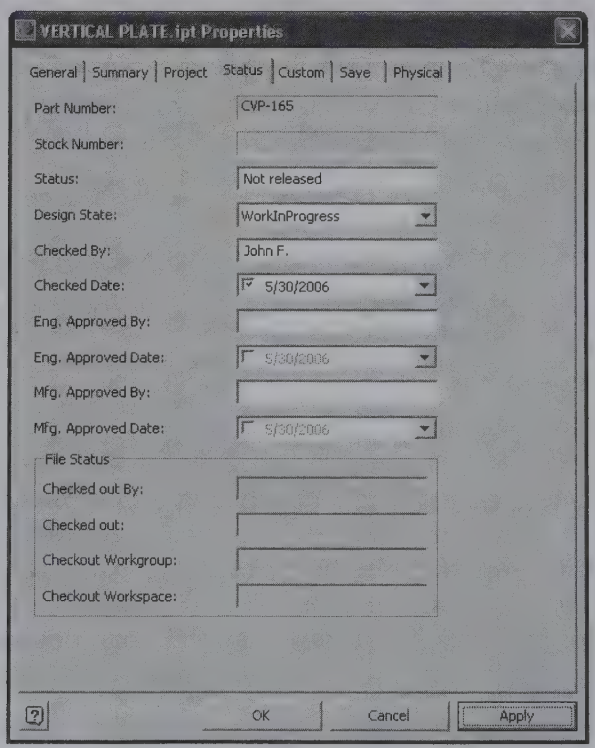


Figure 3-4.
Progress is monitored in the **Status** tab of the **Properties** dialog box.



- **Custom.** This tab, shown in Figure 3-5, allows you to create and delete custom properties to use in the specified file. Custom properties are used to develop any additional user-defined information that may be required to further organize and help control files. Custom properties also help you add information to title blocks, parts lists, reports, and bills of material that is not available from default properties.
  To create a custom property, pick the **Name** edit box and type a property name. Then, select the type of property from the **Type** drop-down list. Next, enter a value, or use the drop-down list, if available, to specify the property value. The particular value corresponds to the property type. Once you complete the custom property information, pick the **Add** button and notice the property shown in the display window. To delete a custom property, pick the property in the display window and select the **Delete** button.
- **Save.** This tab is exactly the same as the **File Save Options** dialog box, and allows you to adjust the save settings of the current files image, displayed in the preview area of the **File Open** dialog box. See Figure 3-6.
- **Physical.** This tab contains a number of options that allow you to set the type of material and adjust and analyze the physical properties of a part or assembly file. See Figure 3-7. You can use the **Physical** tab to show how different materials and dimensions influence the physical and inertial properties of the model.

NOTE



When you select a material in the **Physical** tab of the **Properties** dialog box, the part model style adjusts to the material. The material and color style is set according to the material you select.

Figure 3-5.
If you need to record information that does not match any default property, you can create custom properties in the **Custom** tab.

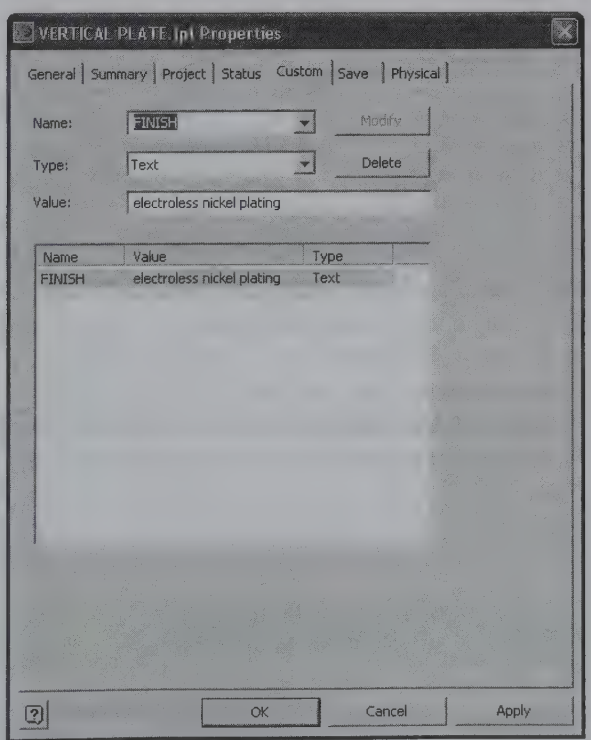


Figure 3-6.
The preview image for each file is controlled in the **Save** tab of the **Properties** dialog box.

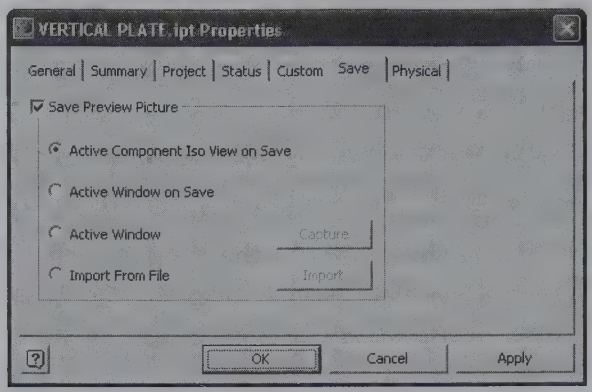
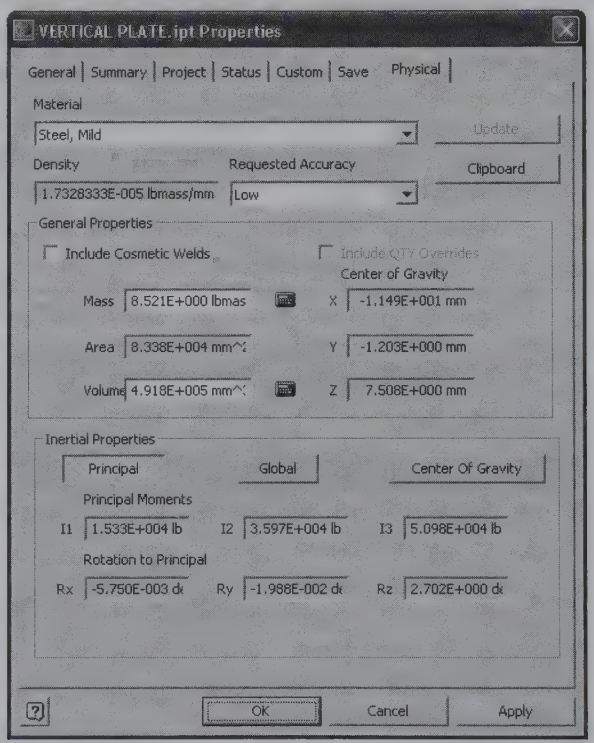


Figure 3-7.
You can input different materials and analyze various physical properties in the **Physical** tab.





Complete the Exercise on the Student CD.

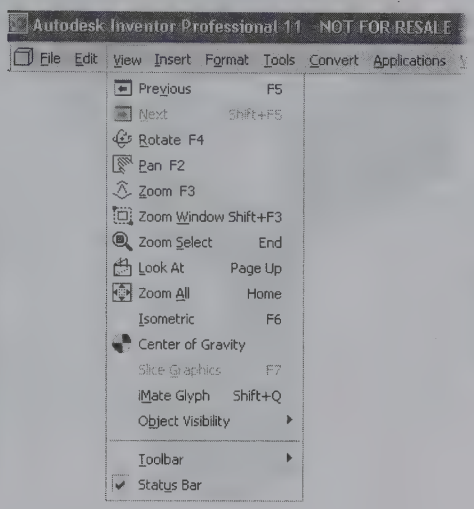
View Options

Inventor view options refer to the actual display characteristics and orientation of a model or drawing, or the display of certain interface items. View options become very important as you progress through the design and documentation process, especially

once you create model features. View tools are used extensively in Inventor primarily because you are working in a three-dimensional world, and you can observe detailed objects, even in a two-dimensional environment. When using Inventor, you are constantly rotating, zooming, and panning as needed to display design characteristics and model orientation, and in order to view areas on which additional sketches, constraints, features, and other modeling items are added.

The **View** pull-down menu, shown in Figure 3-8, is the main source of view options. However, many view tools are available from the **Inventor Standard** toolbar, or in some cases, by right-clicking in the graphics window or on an object in the graphics window.

Figure 3-8.
The **View** pull-down menu contains many options to control the display in the graphics window.



The first option, **Previous**, returns the view to the previous display. You can also access the **Previous View** tool by right-clicking in the graphics window and selecting **Previous View** or by pressing the [F5] key.

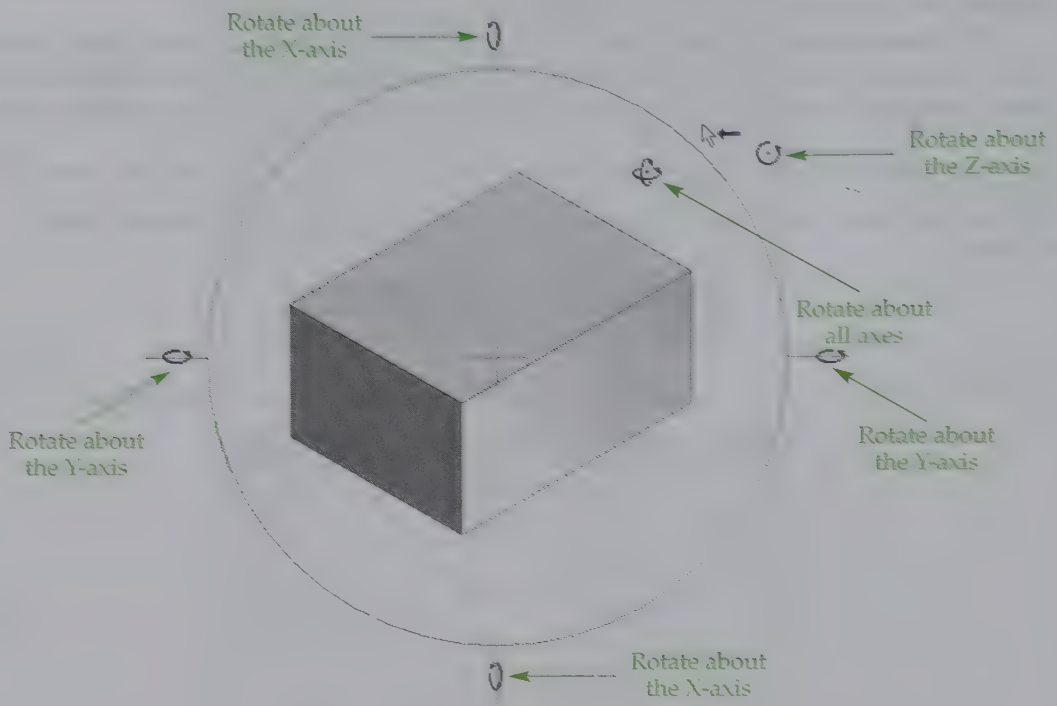
Many times you will need to view or modify various sides of an item. The **Rotate** option allows you to rotate an object and view it from different angles. In addition to the **View** pull-down menu, you can also access the **Rotate** tool by picking the **Rotate** button on the **Inventor Standard** toolbar or by holding down the [F4] key. There are two rotation modes available for viewing an object. When you access the **Rotate** tool, by default, you are in free-rotate mode. See Figure 3-9.

To rotate an object while in free-rotate mode, hold down the left mouse button and drag the rotation icon and object to the desired display. If you choose inside the circle shown in Figure 3-9, you can rotate the object around all axes.

To rotate the object around the Y-axis, move your cursor over one of the horizontal lines, hold down the left mouse button, and drag the rotation icon left or right. To rotate the object around the X-axis, move your cursor over one of the vertical lines, hold down the left mouse button, and drag the rotation icon up or down. To rotate the object around the Z-axis, move your cursor outside of the circle, near one of the quadrants. Then hold down the left mouse button, and drag the rotation icon in a circular pattern. When you are done using the **Rotate** tool, press the [Esc] key, pick outside the rotation area or right-click and choose the **Done** menu option.

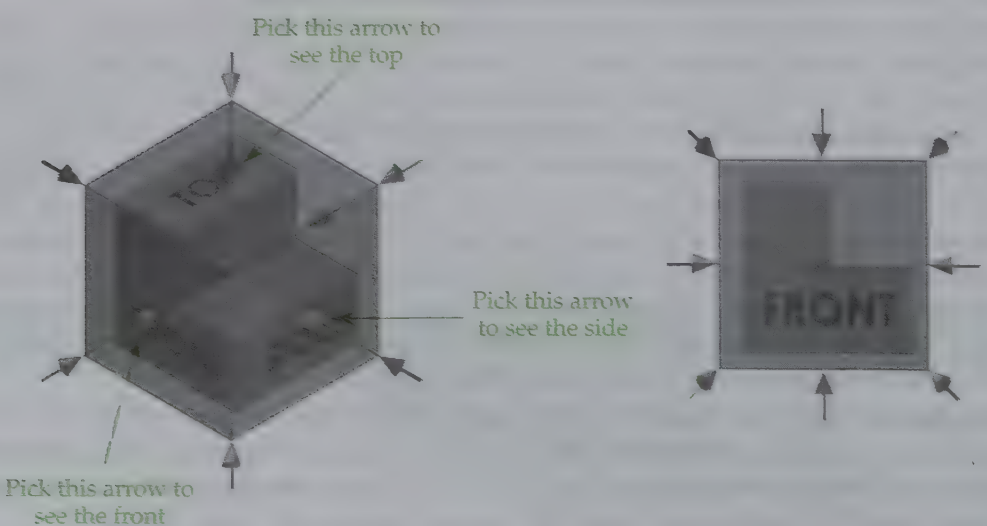
The second mode available for rotating an object is the common-view mode. See Figure 3-10. Access the **Common View** tool by first entering the **Rotate** tool. Then, right-click and select the **Common View [Space]** menu option. To rotate an object while in common-view mode, pick the desired green arrow. Each arrow points to the view it displays.

Figure 3-9.
The position of the cursor determines the method of rotation in the free-rotate mode of the **Rotate** tool.



The **Pan** option lets you reposition objects in the graphics window. To pan, click and hold while moving the **Pan** icon around the graphics window. You can also access the **Pan** tool by pressing and holding the wheel of a wheel mouse, by pressing and holding the [F2] key, or by selecting the **Pan** button on the **Inventor Standard** toolbar.

Figure 3-10.
The common-view mode of the **Rotate** tool allows you to rotate the view to a standard position by picking one of the arrows.



Select **Zoom Window** to zoom into a particular area by creating a window around the area. You can also access this tool by picking the **Zoom Window** button on the **Inventor Standard** toolbar or pressing [Shift]+[F3]. To use the **Zoom Window** option, access the tool, and create a window around the area you would like to zoom in on, by picking a start corner and then picking the opposite diagonal corner. See Figure 3-11.

Use the **Zoom Select** tool to zoom in on a specific object. To use the **Zoom Select** tool, pick the object on which you want to zoom. Pick the **Zoom Selected** button on the **Inventor Standard** toolbar or press the [End] key. See Figure 3-12.

Figure 3-11.
Pick two points or click and drag to create a window around the area to zoom.

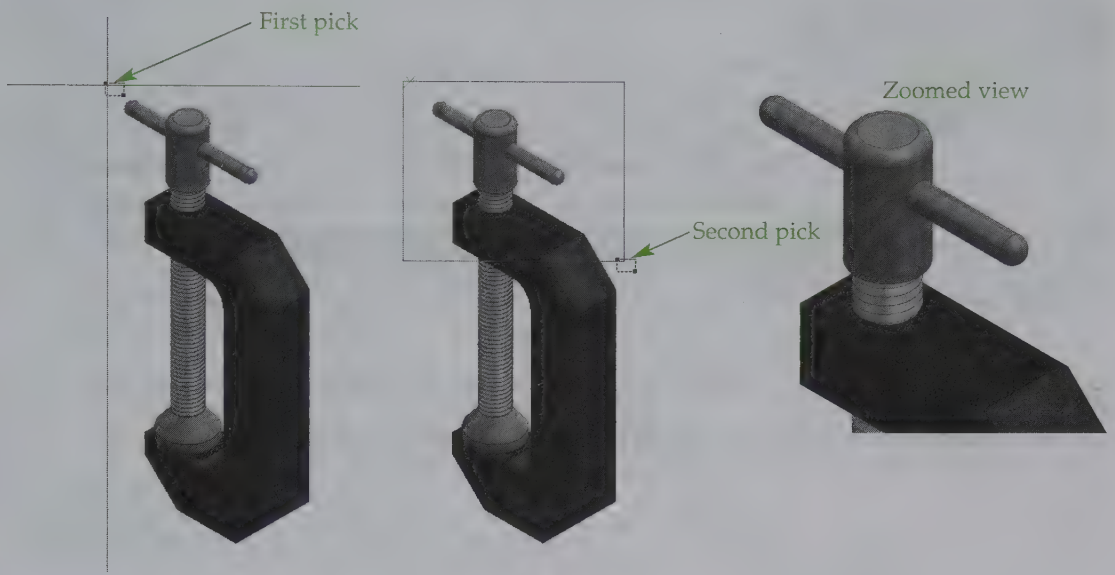
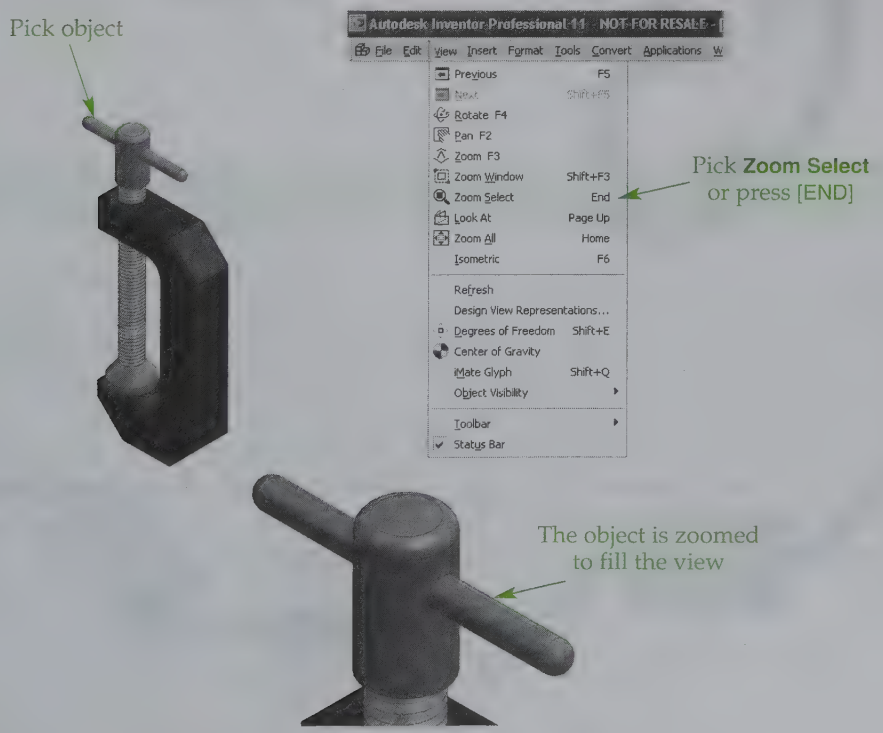


Figure 3-12.
After picking **Zoom Select**, pick an object.



Use the **Look At** tool to rotate the display of a selected face or plane perpendicular to your line of sight, or to display a selected line or edge as horizontal. You can also access the **Look At** tool by picking the **Look At** button on the **Inventor Standard** toolbar, right-clicking on an object and choosing the **Look At** menu option, or pressing the [Page Up] key. See Figure 3-13.

To see all the objects in the graphics window, use the **Zoom All** tool. The **Zoom All** tool is also accessed by picking the **Zoom All** button on the **Inventor Standard** toolbar, pressing the [Home] key, or double-clicking the wheel on a wheel mouse.

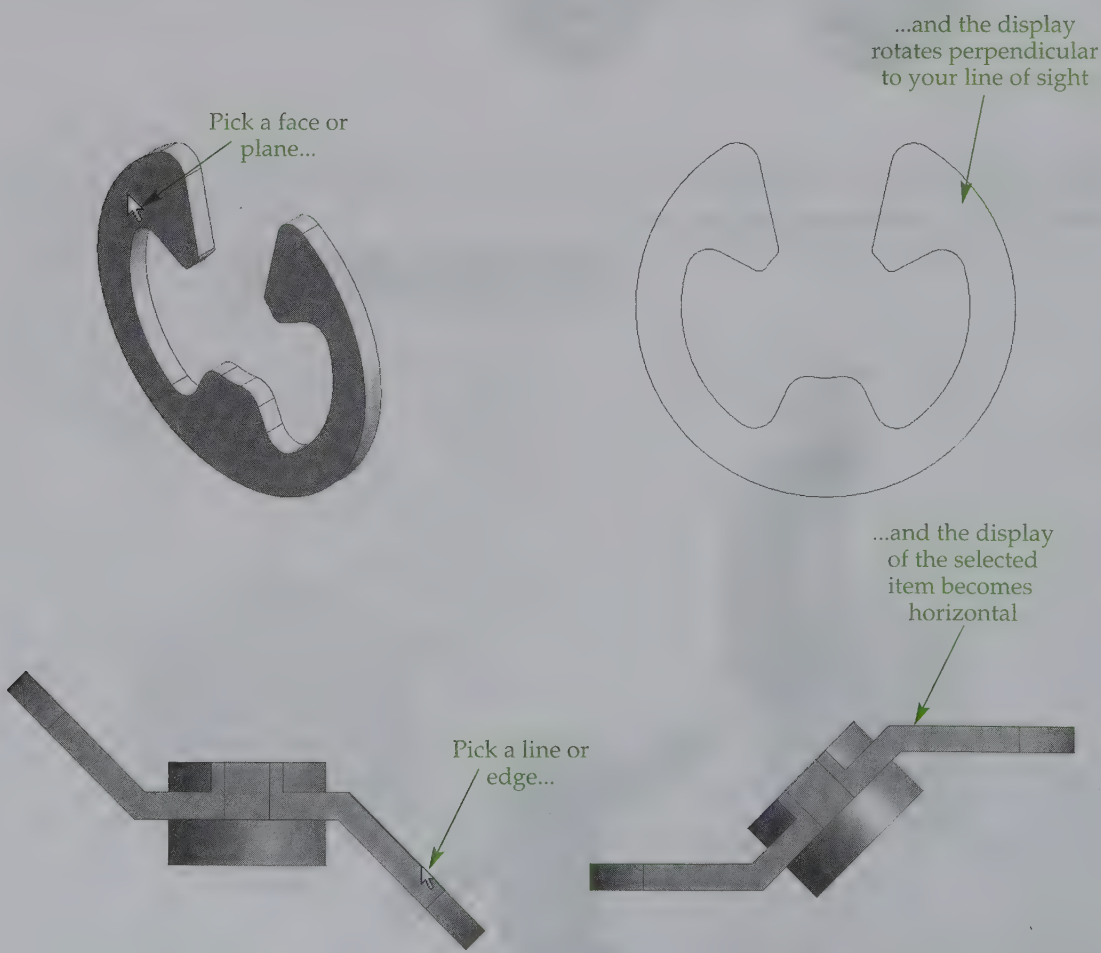
Use the **Isometric** tool to display the isometric view of an object. You can also access the **Isometric** tool by right-clicking and selecting the **Isometric View** menu option, or pressing [F6].

PROFESSIONAL TIP



You can redefine the default isometric view by first rotating the object to the desired display using one of the corner arrows of the **Common View** tool. Then, while still in the **Common View** tool, right-click and choose the **Redefine Isometric** pop-up menu option. If you selected an arrow not pointing to a corner, the **Redefine Isometric** option is grayed out.

Figure 3-13.
Use the **Look At** tool to rotate the view based on a selected plane, line, or edge.



The **Object Visibility** cascading submenu controls the visibility of sketches and categories of work features. These selections are useful when the graphics window becomes cluttered. See Figure 3-14.

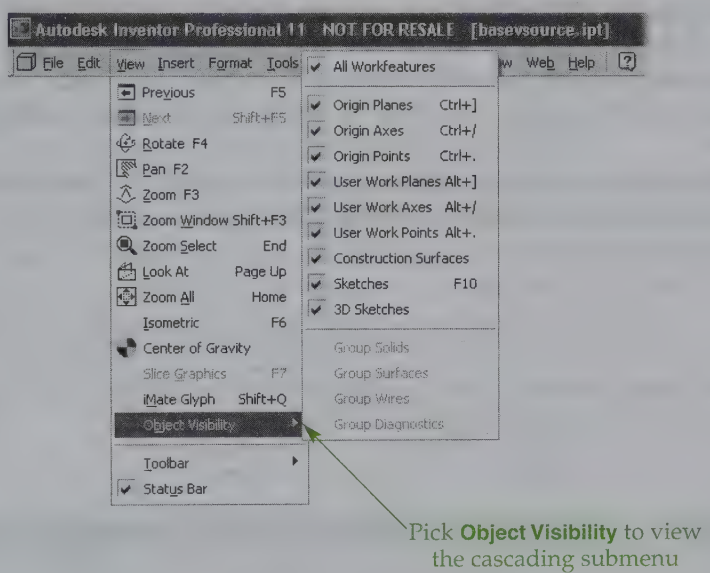
NOTE

The view options previously described rotate the entire graphics space, including the display of the objects. They do not actually change the location of any object.

EXERCISE 3-2

Complete the Exercise on the Student CD.

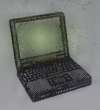
Figure 3-14.
Elements with check marks, in the **Object Visibility** submenu, will be visible.



Styles and Standards

Inventor styles are used in the model work environment to define specific color, lighting, and material characteristics used when creating three-dimensional solid models. For example, if you build a model of a stainless steel bolt you can use a stainless steel color, a specific lighting configuration, and a stainless steel material. There are numerous default styles available with Inventor. If one of these styles is appropriate for your application, it is just a matter of activating the style, which can be done anytime during the design process. If none of the default styles work for your application, create a copy of an existing style that is similar to your requirements, edit the new style as needed, and then save the style to the styles library, so it can be used for future models.

PROFESSIONAL TIP



Styles can be created and modified anytime during the design process. However, you may want to set model styles in your templates, so the same color, lighting, and material is automatically applied each time you develop a certain type of product.

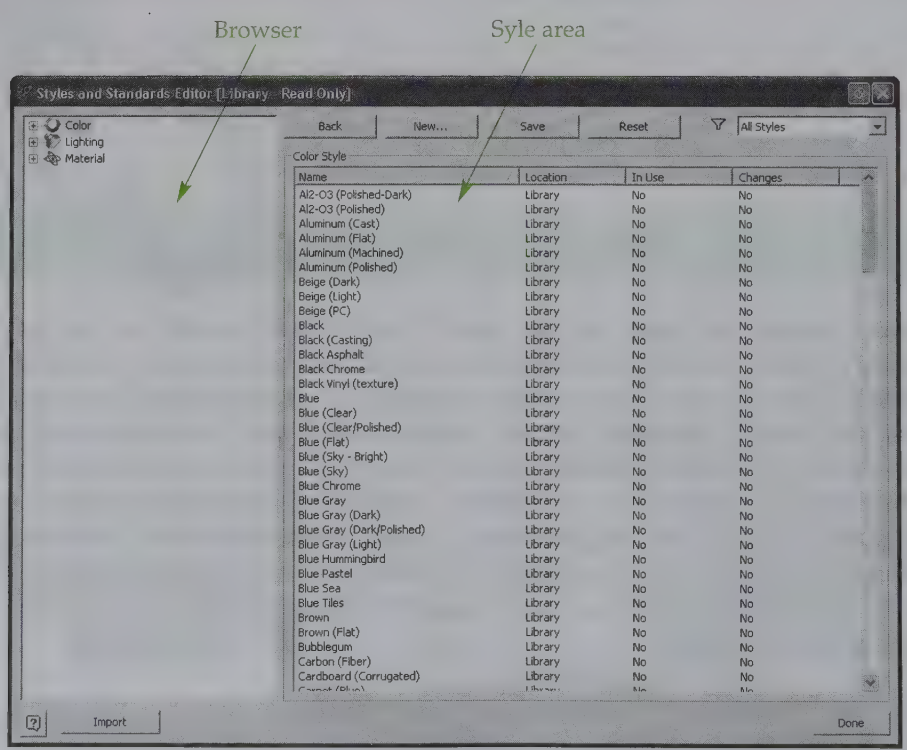
The Styles and Standards Editor

The Inventor **Styles and Standards Editor**, shown in Figure 3-15, is the primary resource for observing and modifying the characteristics of existing color, lighting, and material styles. You can also use the **Styles and Standards Editor** to create new styles and import styles. Pick **Format>Styles Editor...** to access the **Styles and Standards Editor**.

The **Styles and Standards Editor** contains a browser, a style area with multiple options, a filter drop-down list, and a number of buttons. Inside the browser are three parent nodes associated with color, lighting, and material styles. Before you explore the color, lighting, and material styles, you may want to become familiar with the following general items found in the **Styles and Standards Editor**. All of these options function the same whether you are working with color, lighting, or material styles:

- **Back.** Pick this button to return to the previous **Styles and Standards Editor** display.
- **New.** Select this button to create a copy of the selected standard. When you pick the **New** button, the **New Style Name** dialog box is displayed, which allows you to create a new style.

Figure 3-15.
The **Styles and Standards Editor** controls color, lighting, and material styles. Notice the number of colors listed in the style area.



- **Save.** If you edit style settings, you must pick the **Save** button to keep the changes. If you do not save after making a change, you are prompted to do so before continuing.
- **Reset.** Returns style changes to the default settings.

NOTE

Saving changes using the **Save** button only saves the changes in the current file, and does not save changes to the style library.

- **Filter.** Use this drop-down list to display only certain styles in the **Styles and Standards Editor**.
- **Import.** Pick this button to import style definition files.
- **Done.** Select this button when you are finished using the **Styles and Standards Editor**, or pick the **Close** button in the upper-right corner.

When you select the **Color**, **Lighting**, or **Material** parent nodes of the **Styles and Standards Editor** browser, the styles are listed in a table format, such as the **Color Style** area shown in Figure 3-15. The titles of the columns, regardless of the parent node selected, are always **Name**, **Location**, **In Use**, and **Changes**. The name of each style in the **Name** column should be descriptive. Some names contain details in parentheses, such as transparency amounts, texture information, and color descriptions.

The **In Use** column identifies whether or not a style is used in the current file. If a style is active or is used in the file, the **In Use** description is **Yes**.

The status of the style is tracked in the **Changes** column. Initially, all styles are listed with No changes. When you edit a style and do not save the changes to the style library, Local is shown. If you edit a style and save the changes to the style library, Library is shown. If a style has been edited in the current file, and the style library is modified, resulting in different values in both places, Both is displayed.

NOTE

You cannot update or save styles to the styles library if the active project **Use Styles Library** definition is set to **No**. You cannot save styles to the styles library if the active project **Use Styles Library** definition is set to **Read Only**.

Activating Styles

An active style is the style that is currently used, or turned on, in the model. For example, if you are working on an aluminum part that is painted red, you may want to activate a red color, a certain lighting style, and an aluminum material. In addition to using the **Styles and Standards Editor** you can activate color, lighting, and material styles from other areas of Inventor.

A color style can be activated by choosing a color from the **Color** drop-down list of the **Inventor Standard** toolbar. See Figure 3-16. A lighting style and a material style can be activated from the **Standard** tab of the **Document Settings** dialog box, shown in Figure 3-17. You can access the **Document Settings** dialog box by selecting the **Document Settings** option from the **Tools** pull-down menu, or by picking the **Active Standard...** option from the **Format** pull-down menu. Select the desired styles from the **Active Lighting Style** and **Material** drop-down lists. A material style can be activated by selecting the desired material from the **Material** drop-down list in the **Physical** tab of the **Properties** dialog box.

Figure 3-16.
Select a color for the part from the **Color** drop-down list in the **Standard** toolbar.

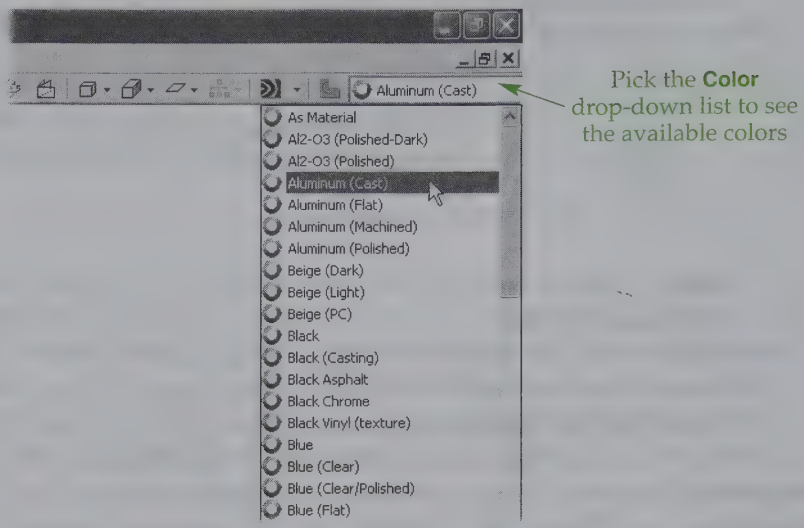
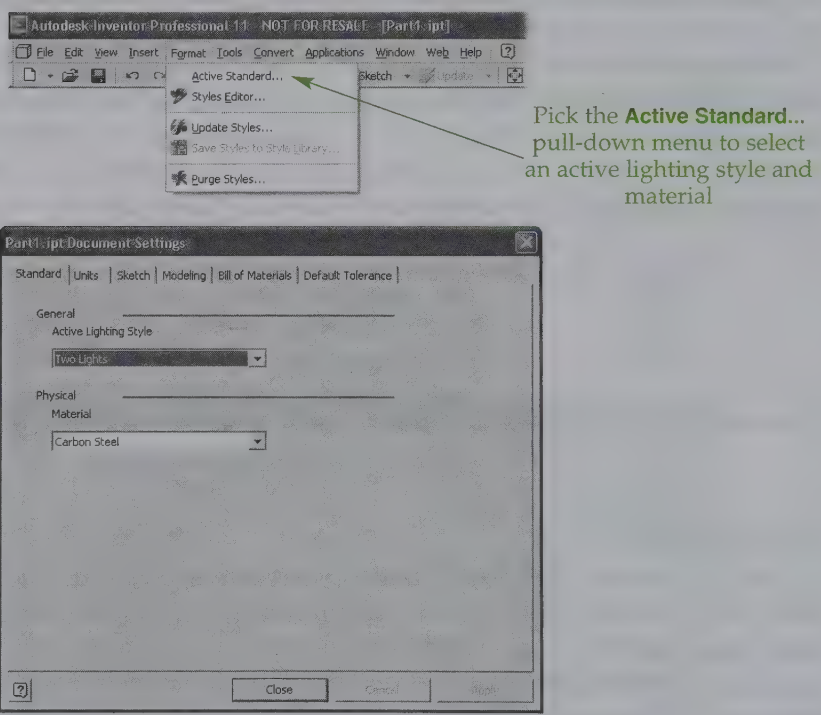


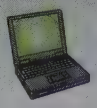
Figure 3-17.
Open the **Document Settings** dialog box to select an active lighting style and material.



Working with Color Styles

Color styles are used to define the color of a part model. The default color is gray, but there are many different colors you can use, including texture options that make models look very realistic. Color selection varies depending on the model and the design requirements. You may choose to use the default color throughout the design process, or you may want to assign a specific color to each part in an assembly for quick identification. You can select a color that matches the part's material, such as a copper color and a copper material. Otherwise, you can apply a color that is different from the material.

PROFESSIONAL TIP



When applying colors and materials, do not apply a color that describes a material that is different from the part's material. For example, confusion could arise if you apply a color called Aluminum (flat) to a part that has a material setting of Stainless Steel. Instead, apply a specific color called Metal-Steel (Stainless Brushed) or a vague color, such as Gray.

When you expand the **Color** node in the **Styles and Standards Editor** browser, all the available color styles are listed as child nodes. See Figure 3-18. Picking a specific color displays the color settings, and the following areas.

- **Colors.** This area contains four color pads that show the current diffuse, emissive, specular, and ambient characteristics of the selected color. Each of these characteristics can be changed by picking on its pad. See Figure 3-19.
- **Appearance.** This section allows you to modify the shiny and opaque characteristics of a part surface in reference to the selected color. To change the appearance of a color, move the **Shiny** and **Opaque** slide bars to the desired setting.

Figure 3-18.
Picking a color in the browser or the style area presents the settings for that color.

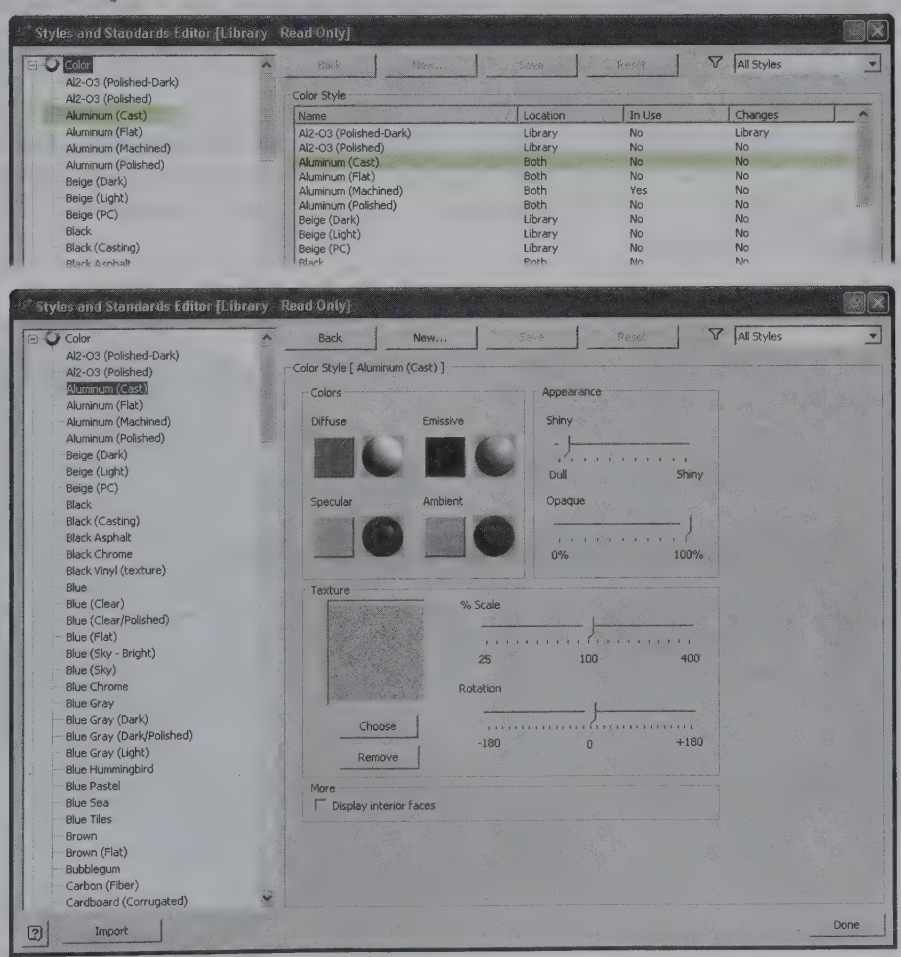
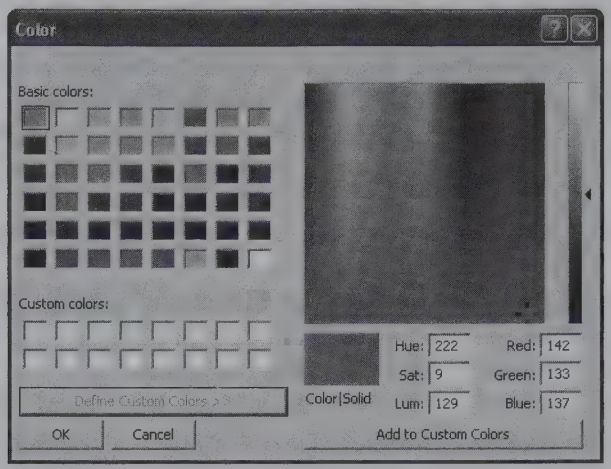
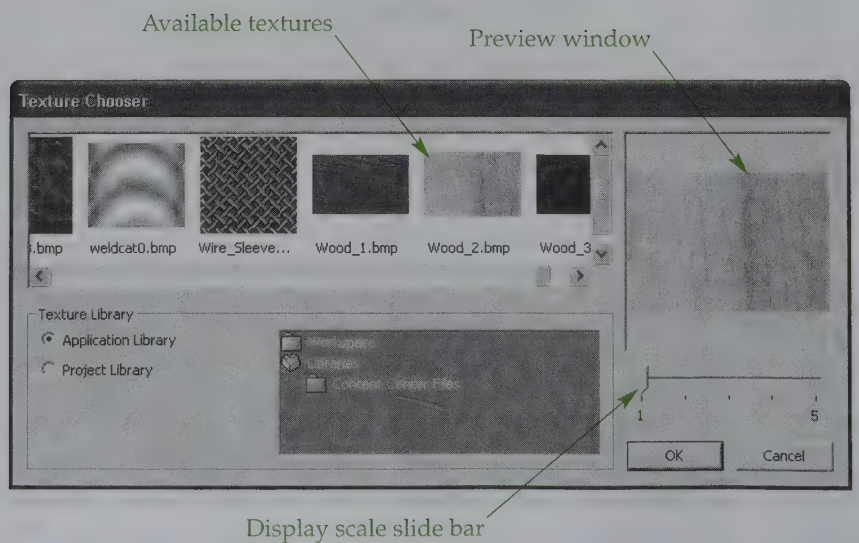


Figure 3-19.
The **Color** dialog box is where basic colors are listed and custom colors can be created.



- **Texture.** This section contains options that allow you to use image files to add texture to the surface of a part, creating a more realistic looking model, or to represent a commercial product, such as a screen. The image files supplied with Inventor include wood grain, various metal finishes, and concrete.
  Pick the **Choose** button to access the **Texture Chooser** dialog box. See Figure 3-20. You can access standard textures by picking the **Application Library** radio button, or you can use project textures you create by picking the **Project Library** radio button. The texture you select from the list is displayed in the preview area. To view the selected texture using a different scale, use the **Scale** slide bar. Changing the texture scale while in the **Texture Chooser** dialog box is for demonstration purposes only and does not actually modify the texture scale. When you are sure you have chosen the correct texture, pick the **OK** button to add the texture to the specified color. Pick the **Remove** button if you decide not to use the selected texture in addition to the color.

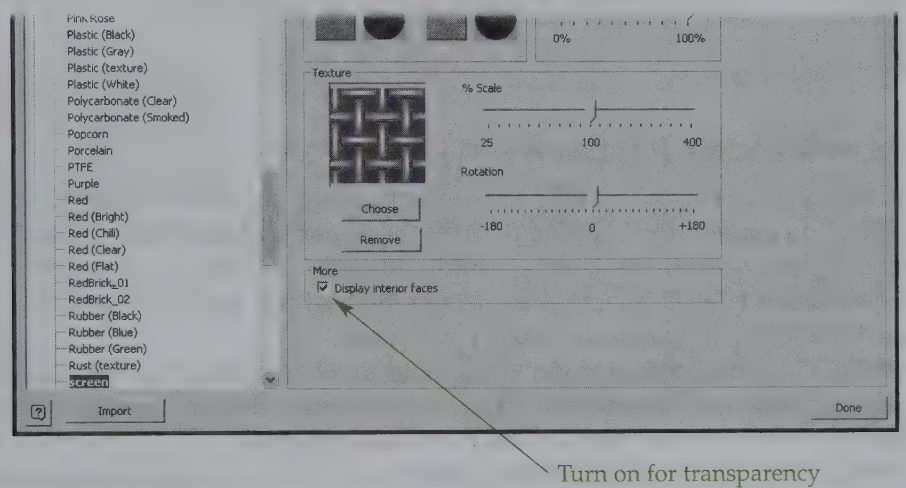
Figure 3-20.
After scrolling through the textures, pick one to see it in the preview window in the **Texture Chooser** dialog box.

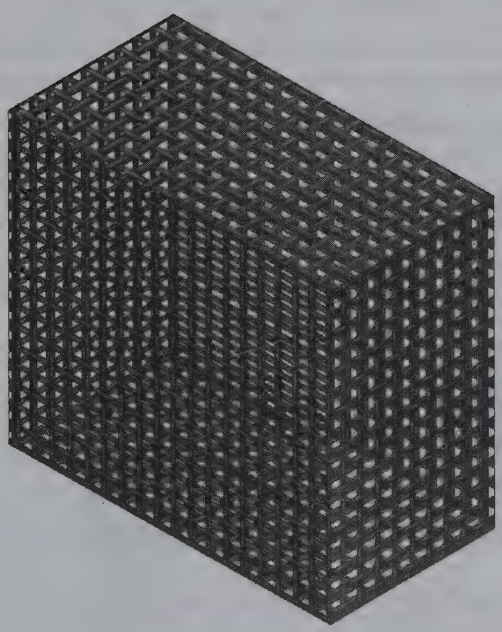


After selecting a texture, click and drag the **Styles and Standards Editor** to a position that allows you to see the part and access the **% Scale** and **Rotation** slide bars. Move the **% Scale** slide bar to adjust the scale of the texture display. Move the **Rotation** slide bar to modify the angle at which the texture is displayed in the model. For example, if you use a texture with horizontal lines, such as a screen, you can increase or decrease the rotation to view the screen at a specific angle, including vertical.

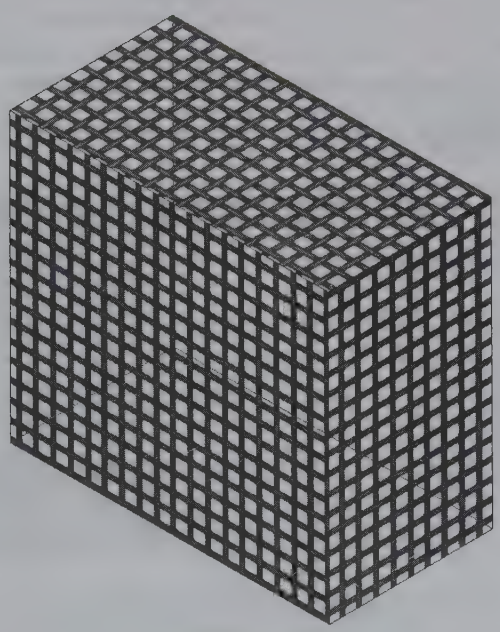
- **More.** This area contains the **Display interior faces** checkbox. When checked, the areas designated transparent of some textures are made transparent in the graphics window. For example, the Inventor textures representing screens are transparent just as a real screen. See Figure 3-21.

Figure 3-21.
Compare the displays of the textured models when the **Display interior faces** checkbox is on and off.





Display interior faces
is checked



Display interior faces
is not checked

NOTE

Typically when modifying colors, and especially when you specify a color texture, it is a good idea to create a new color instead of adjusting the default color.

PROFESSIONAL TIP

Multiple texture options are available in the **Content Center** folder, but you can create your own textures from .bmp, .gif, .jpg or .jpeg, .png, or .tiff images and store them in image libraries assigned to a project.

EXERCISE 3-3

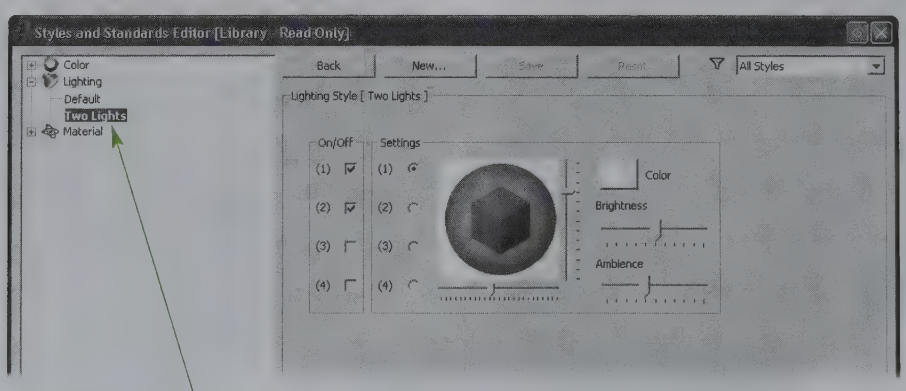
Complete the Exercise on the Student CD.

Working with Lighting Styles

Lighting styles are used to define various lighting characteristics applied to the model. When you expand the **Lighting** node in the **Styles and Standards Editor** browser, the available lighting styles are listed as child nodes. By default, two lighting styles are available: Default and Two Lights. Picking a specific lighting style displays the lighting settings and the following areas, as shown in Figure 3-22.

- **On/Off.** This area contains four checkboxes that allow you to activate or suppress lighting settings. When you activate more than one of the checkboxes, the lighting effects of each setting are combined together.
- **Settings.** This area controls lighting preferences. The four radio buttons correspond to the **On/Off** checkboxes and allow you to create different lighting options for the same style. Pick the desired radio button and apply the following settings:

Figure 3-22.
Picking a lighting style reveals the number of lights and the settings for each light.



The **Lighting** node is expanded and the **Two Lights** lighting style is selected

- **Lighting position slide bars.** The vertical lighting position slide bar controls the amount of light provided above and below the model. The horizontal lighting position slide bar controls the amount of light provided left and right of the model.
- **Color.** Pick this pad to specify the color of the light used to illuminate the part.
- **Brightness.** This slide bar allows you to change the brightness of the light source for all settings. If you would like to change the brightness for just one setting, you need to change the light source color.
- **Ambience.** This slide bar controls the contrast of the ambient, or overall, lighting.

NOTE



If you change the light source color, all model and material colors reflect the new color. Typically when modifying lighting, it is a good idea to create a new lighting style instead of adjusting the default styles.

PROFESSIONAL TIP



When changing lighting options, you may want to have a model in the graphics window to instantly see the changes made by your selections.



Complete the Exercise on the Student CD.

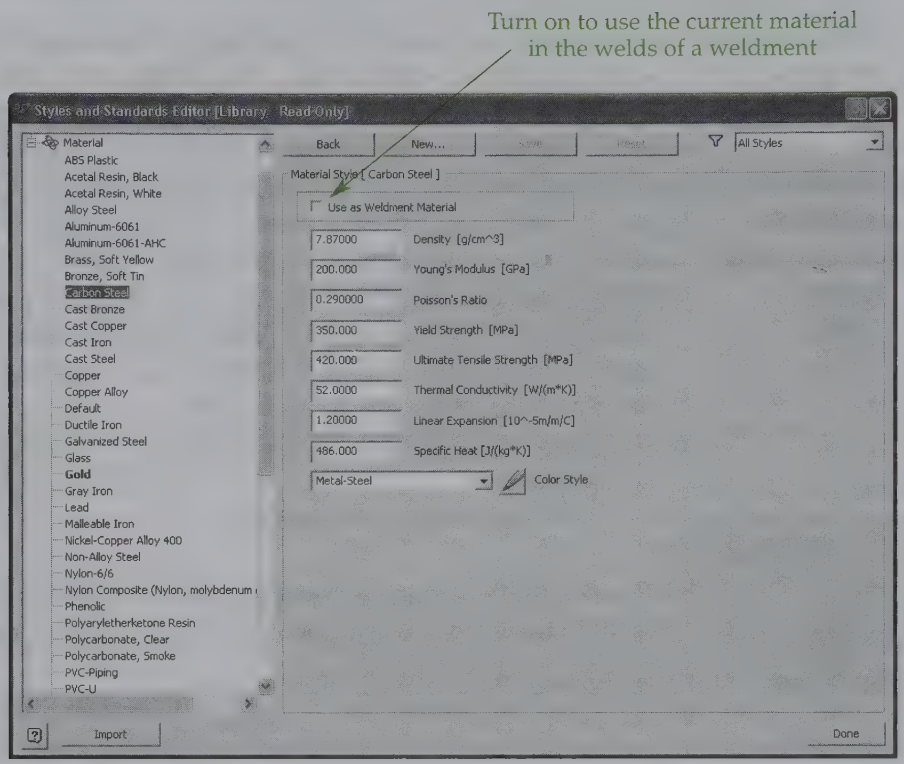
Working with Material Styles

Inventor allows you to specify the type of material used to create a part. There are a large number of available material styles ranging from lead to ultra high molecular weight polyethylene. As with other styles, you can also create your own materials. The material you select defines the actual part material, and is intended to be a virtual representation of the real product material. As you noticed in the **Physical** tab of the **Properties** dialog box, model materials contain physical properties that can be used to influence the physical and inertial characteristics of the model. Additionally, when you select a model material, the material can be referenced throughout the design and documentation process in items such as a bill of materials.

When you expand the **Material** node in the **Styles and Standards Editor**, all the available material styles are listed as child nodes. Picking a specific material displays the material settings, and the following areas, as shown in Figure 3-23.

- **Use as Weldment Material.** Weldments are assemblies of parts that are held together using welds. In many cases, the same material used to create a part is used to weld parts together. If this is the situation, pick the **Use as Weldment Material** checkbox to have the material available when generating a weldment model.
- **Properties.** A variety of properties are associated with each type of material. Each property contains an edit box that allows you to adjust the physical properties of the selected material. These properties are used to calculate the physical properties of a part. To modify an existing property, highlight the property value and enter a new value in the edit box.

Figure 3-23.
The physical properties of a material can be seen and adjusted by picking the material in the browser. These properties can also be applied to any welds in a weldment.



- **Color Style.** Each material is assigned a default color style. Material color styles are exactly the same as the color styles. Materials reference an existing color style, which means that color styles become sub-styles when working with materials. For example, the Lead material style is assigned the Metal-Lead color style. You can assign a different default color to a material by selecting a color from the **Color** drop-down list. You can also access the **Color Style** area of the selected color by picking the **Color Style** button. You should only change the default material color if you want that color used every time the material is used.

NOTE

If you want to use the color defined by the selected material, do not pick a different color, and be sure that the **As Material** option of the **Inventor Standard** toolbar, **Color** drop-down list, is selected.

PROFESSIONAL TIP

You may want a part to be made from a specific material, but use a different color in order to distinguish parts from each other in an assembly or replicate painting the part.

EXERCISE 3-5

Complete the Exercise on the Student CD.

Updating Styles

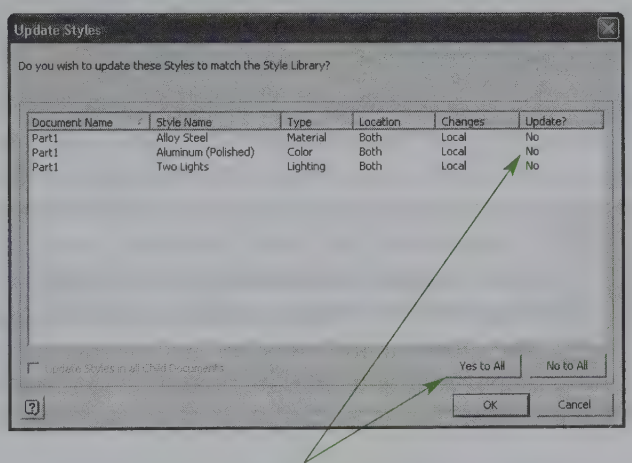
As previously mentioned, the purpose for updating styles is to ensure that your styles are associated with the styles library. Only styles that have not been saved to the styles library can be updated. You can update individual styles using the **Style Update** option of the **Styles and Standards Editor**. However, in some cases many styles may require updating. If this is the case, pick **Format>Update Styles...** to access the **Update Styles** dialog box. See Figure 3-24. The **Update Styles** dialog box contains information regarding the document name, style name, type, location, and changes. If you do not want to update any of the styles, pick the **No to All** button. If you want to update all the styles, pick the **Yes to All** button. You also have the option of selecting specific styles to update, by picking either **No** or **Yes** from the **Update?** list. In addition, if you want to automatically update the styles in files that reference the parent file (such as the way an assembly references its components), pick the **Update Styles in all Child Documents** checkbox.

NOTE



Updating styles according to the styles library eliminates any style information that has not been saved to the style library.

Figure 3-24.
The **Update Styles** dialog box lists all of the styles that have not been saved to the styles library.

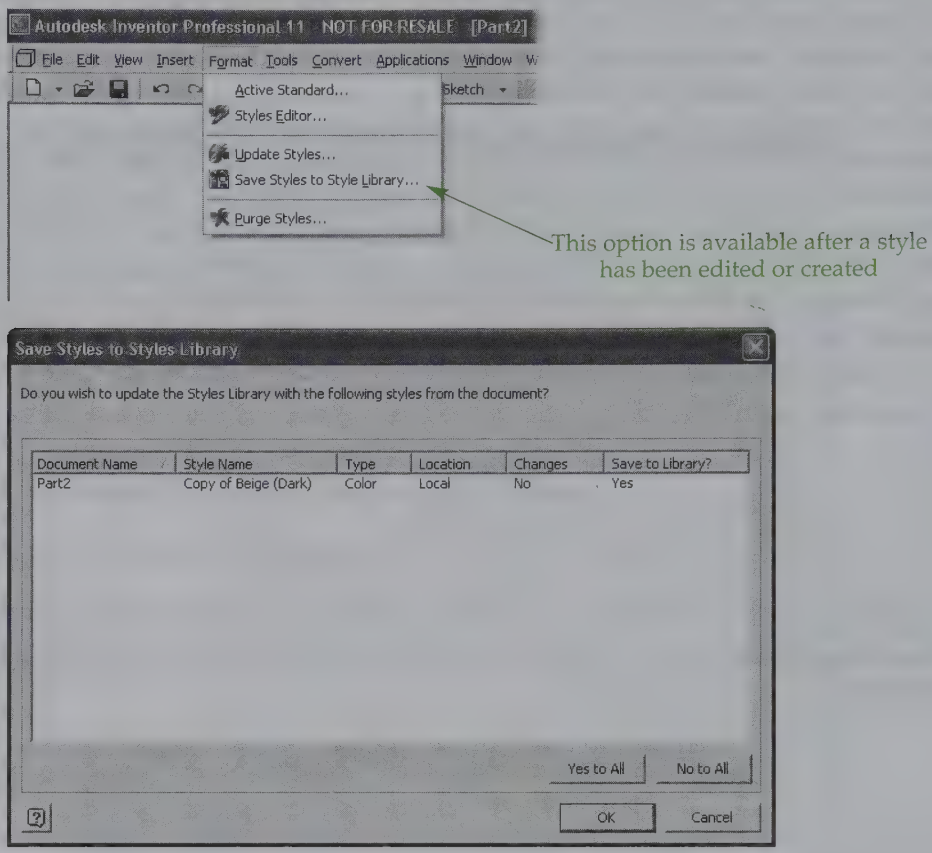


Select the **Yes to All** button to change the **Update?** column to **Yes**

Saving Styles to the Styles Library

When you modify a style or create a new style, you must save the changes to the style library in order for the style to be effectively used in other documents. If you do not save style changes, then only the current file contains the style settings. Once you edit or create a new style, save the modifications of an individual style to the styles library by selecting the **Save To Styles Library** option of the **Styles and Standards Editor**. Another option is to save all style modifications at once by picking **Format>Save Styles To Styles Library**. This opens the **Save Styles To Styles Library** dialog box. See Figure 3-25. The **Save Styles To Styles Library** dialog box contains information regarding the file name, style, name and type, and the **Location** and **Changes** lists. If you do not want to save any of the styles, pick the **No to All** button. If you want to save all the styles to the styles library, pick the **Yes to All** button. You also have the option of selecting specific styles to save, by picking either **No** or **Yes** from the **Save to Library?** list.

Figure 3-25.
The **Save Styles To Styles Library** dialog box lists the new or modified styles.



PROFESSIONAL TIP



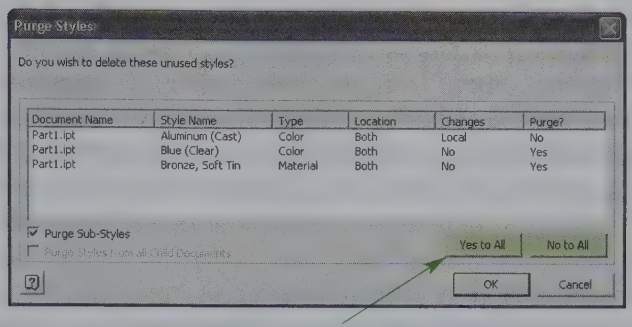
Use caution when editing and saving styles to the styles library. If you think you may want to access default styles in the future, it is always best to create a copy of a style, edit the copy, and then save the new style to the styles library.

Purging Styles

One of the main reasons styles are used in Inventor is to provide a method of storing all of the information regarding styles in a place other than the part, drawing, and assembly files. Styles are copied to these files as needed and therefore add to the size of the files. Any unused styles can be removed from a file through purging. Purging styles allows Inventor to function better, especially when dealing with large, multi-component assemblies.

You can purge individual styles using the **Purge Style** and **Purge Style and Sub-Style** options of the **Styles and Standards Editor**. Styles are also purged in the **Purge Styles** dialog box by picking **Format>Purge Styles...** See Figure 3-26. The **Purge Styles** dialog box contains information regarding the file name, style, name and type, and the **Location** and **Changes** lists. If you do not want to purge any of the styles, pick the **No to All** button. If you want to purge all the styles, pick the **Yes to All** button. You

Figure 3-26.
Use the **Purge Styles** dialog box to remove unused styles.



also have the option of selecting specific styles to purge, by picking either **No** or **Yes** from the **Purge?** list. In addition, if you want to automatically purge the styles in files that reference the parent file (such as an assembly file references parts files), pick the **Update Styles in all Child Documents** checkbox, and if you want to automatically purge any sub-styles associated with styles, pick the **Purge Sub-Styles** checkbox.

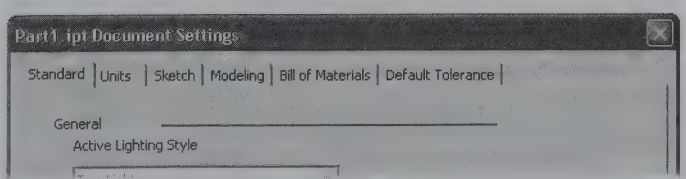
Document Settings

Inventor document settings control several work environment options associated with the current file. You should be familiar with many of these settings before you create a model. Additionally, document settings can be customized in template files so that specific options are already set each time you start a new model.

To access document settings and the **Document Settings** dialog box, pick **Tools>Document Settings...** The **Document Settings** dialog box includes tabs, shown in Figure 3-27, labeled **Standard**, **Units**, **Sketch**, **Modeling**, **Bill Of Materials**, and **Default Tolerance**.

The **Standard** tab, where the active material and lighting are set, was covered in the *Styles and Standards* section of this chapter. The other tabs will be described when applicable.

Figure 3-27.
Each of the tabs in the **Document Settings** dialog box will be covered when appropriate.



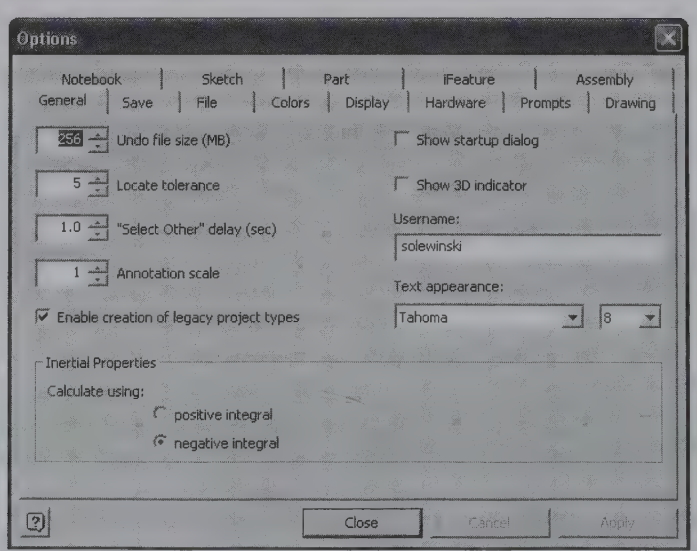
Application Options

Inventor application options are contained in the **Options** dialog box, which you access by picking **Tools>Application Options...** Inventor application options are different

from document settings, because they are general program options, and are not specific to any particular file. However, many application options do help configure work environments. Though application options are not set in templates, because they apply to the entire program, you should become familiar with the following tabs before you begin work on a model:

- **General.** This tab, shown in Figure 3-28, sets various general Inventor options. These choices are described as follows:
  - **Undo file size (MB).** Use this edit box to set the amount of disk space used for the **Undo** option. Pick or right-click inside the edit box to type in a number, or use the up and down arrows to increase or decrease the number of megabytes.
  - **Locate tolerance.** This edit box controls how far away from an object you can pick and still select the object. The distance is defined as the number of pixels from 1 to 10. You can select or double-click inside the edit box and type a distance, or pick the up or down arrows to increase or decrease the distance.
  - **"Select Other" delay (sec).** Use this edit box to specify the number of seconds you can hover over model geometry before the **Select Other** tool is shown.
  - **Annotation scale.** Use this edit box to set the displayed size of all annotation components in the graphics window, including the size of the 3D indicator, sketch dimensions, work points, selection points, and icons. The annotation scale you define in the **General** tab of the **Options** dialog box does not affect the actual size of any model parameters or drawing annotations, and represents only an option for defining certain display properties. The default normal annotation size is 1, but you can choose any scale between 0.2 and 5.0.
  - **Enable creation of legacy project types.** Pick this check box if you have a need to access other project types, related to earlier Inventor versions, or if you do not have Vault loaded and want to use the collaborative functions of Inventor projects. For most application the **Enable creation of legacy project types** check box does not require selection.
  - **Show Startup Dialog.** This check box specifies whether the Inventor **Open** dialog box opens automatically. If the check box is clear, the Inventor **Open** dialog box does not automatically open when Inventor is launched.

Figure 3-28.
The **Options** dialog box, **General** tab.



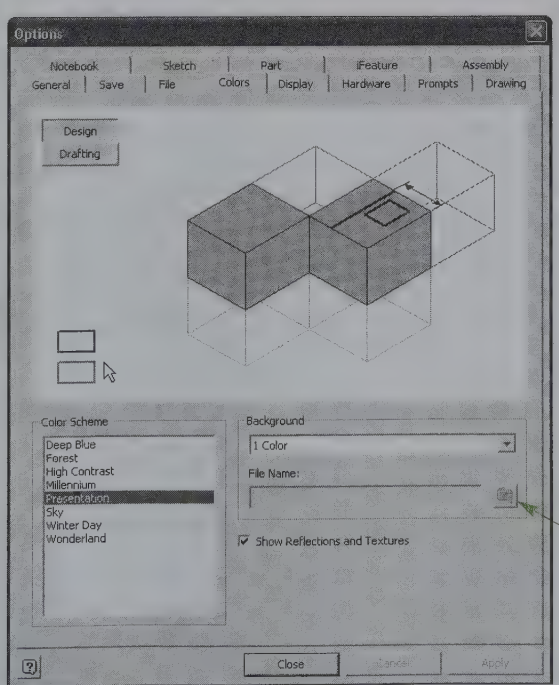
- **Show 3D Indicator.** An XYZ axis icon shown in Figure 3-29, is displayed in the graphics window of every Inventor three-dimensional work environment. The 3D indicator is an important feature that allows you to identify the location and orientation of a model in three-dimensional space. The X-axis is portrayed as a red arrow, the Y-axis as a green arrow, and the Z-axis as a blue arrow. The **Show 3D Indicator** check box is used to turn the 3D indicator on and off. If the check box is clear, the 3D indicator is not displayed in the graphics window.
- **Username.** Use this edit box to enter your username. This name is applied to a number of tools and options.
- **Text appearance.** This area contains two drop-down lists that allow you to control the appearance of the text used in a number of Inventor interface items such as edit boxes, title bars, the panel bar, the browser bar, and sketch dimensions. Use the **Text font** drop-down list to select the desired font, and use the **Text size** drop-down list to identify the text height. As with the annotation scale, the text appearance settings do not affect text styles used in drawings and are only meant to define work environment display characteristics.
- **Inertial Properties.** This area contains a **Negative Integral** and a **Positive Integral** radio button, one of which can be selected to define how inertial properties are defined in the **Physical** tab of the **Properties** dialog box.

- **Colors.** This tab, shown in Figure 3-30, controls the background color displayed in the graphics window. Inside this tab you can specify the background color of the modeling environment using the **Design** button, and the drawing environment using the **Drawing** button. You can choose to set a single color for the background

Figure 3-29.
The Inventor 3D indicator.



Figure 3-30.
The **Options** dialog box, **Colors** tab.



display by selecting **1 Color** from the **Background** drop-down list and then picking the desired color from the **Color Scheme** list box. Another option is to select **Gradient** from the **Background** drop-down list to display the specified background color scheme as a multi-toned gradient. You can also choose to use a bitmap (.bmp) image for the background. To display an image, first select **Background Image** from the **Background** drop-down list. Then, pick the **Browse** button to access the **Open** dialog, allowing you to choose an image file.

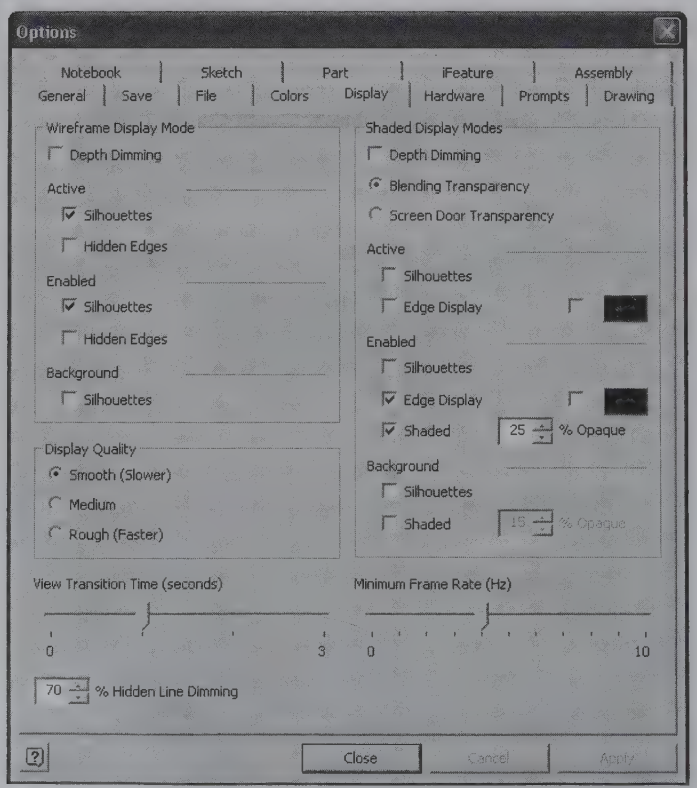
PROFESSIONAL TIP



Inventor contains a number of bitmap images that can be used as a background. These files are located in the following path: Autodesk/Inventor 11/Backgrounds. You can also save your own bitmap images to use as a background.

- **Display.** This tab, shown in Figure 3-31, controls wireframe and shaded display modes. The following areas and options are located inside this tab:
  - **Wireframe Display Mode.** This area specifies the wireframe display options for a model.
  - **Display Quality.** This area controls the model display resolution. Consequently, the regeneration time is also adjusted here.
  - **Shaded Display Modes.** This area specifies the shaded display options for a model.
  - **View Transition Time (seconds).** Move the slide bar left or right to control the time it takes to change between various views, such as zooming in and out,

Figure 3-31.
The **Options** dialog box, **Display** tab.

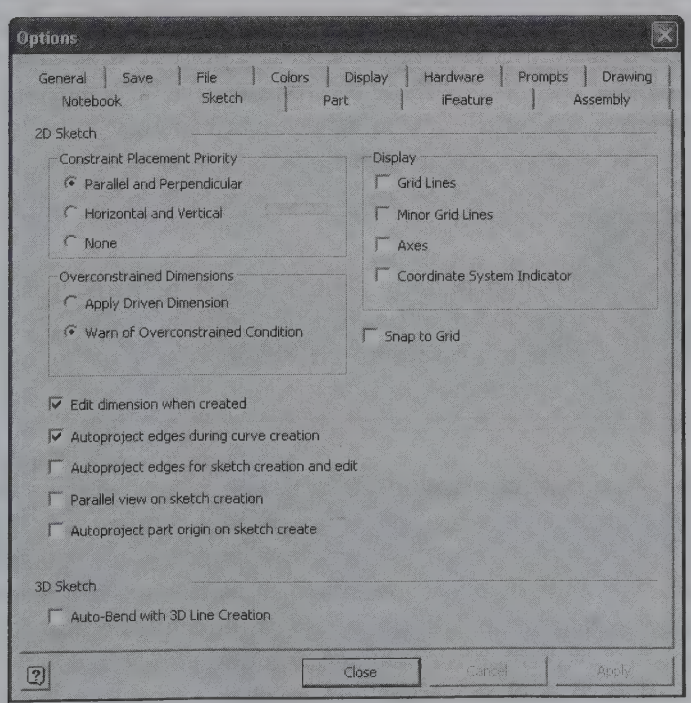


panning, and looking at a specific object. Seconds can be any number from 0 to 3. Setting transition time to 0 seconds allows changing between views to happen almost instantaneously, while setting transition time to 3 seconds allows for the longest time between view transitions.

- **Minimum Frame Rate (Hz).** Move the slide bar left or right to control the rate at which frames regenerate between various views, such as zooming in and out, panning, and rotating. Frame generation can be specified as any number from 0 to 10. Setting minimum frame rate to 0 allows all the frames with no time limit, while setting mini-mum frame rate to 10 allows at least 10 frames do be developed per second. Although you can adjust these settings, generally the frame rate is adjusted much faster than what you specify.
- **% Hidden Line Dimming.** This edit box allows you to control the dimming percentage for hidden edges. The **% Hidden Line Dimming** edit box is available only when at least one of the **Hidden Edges** check boxes is picked. You can enter a new percentage or scroll through available percentages using the up and down arrows.

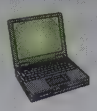
- **Sketch.** This tab, shown in Figure 3-32, allows you to specify two-dimensional and three-dimensional sketch settings. The following describes the Sketch tab options:
  - **Constraint Placement Priority.** Inventor has the ability to automatically create some designated constraints if applied. This area allows you to give constraint priority to either parallel and perpendicular constraints or horizontal and vertical constraints, by selecting the appropriate radio button.
  - **Display.** This area allows you to turn on or off grid lines, axes, and the coordinate system indicator, by selecting or deselecting the corresponding check boxes.
- **Overconstrained Dimensions.** This section houses two radio buttons that allow you to apply over-constrained dimensions to a sketch or warn you if a sketch is over-constrained. If you select the **Apply Driven Dimension** radio

Figure 3-32.
The **Options** dialog box, **Sketch** tab.



button, a nonparametric, driven dimension is created. If you select the **Warn of Overconstrained Condition** radio button, a warning message is displayed if a sketch becomes over-constrained.

PROFESSIONAL TIP



Usually, it is most helpful to pick the radio button **Warn of Overconstrained Condition** to ensure a sketch does not become overconstrained and nonparametric. However, if you want to add several reference dimensions, you can pick the **Apply Driven Dimension** radio button; so that you do not have to constantly deal with the overconstrained warning.

- **Snap to Grid.** Picking this check box allows you to snap directly to a grid point when sketching geometry or locating a point.
- **Edit dimension when created.** If you pick this check box, an **Edit Dimension** dialog box appears. See Figure 3-33. This is a very valuable option that allows you to enter a specific value for the object when you place a dimension.
- **Autoproject edges during curve creation.** Picking this check box allows you to project the edge of an existing feature onto the sketch plane while using sketch tools.
- **Autoproject edges for sketch creation and edit.** Selecting this check box automatically projects referenced feature geometry edges onto a new sketch plane.
- **Autoproject part origin on sketch create.** Choose this check box to automatically project the center point onto a new sketch plane.
- **Auto-Bend with 3D Line Creation.** When creating lines in a 3D sketch environment, selecting this check box creates an arc or corner bend at the line intersection.

- **Part.** This tab, shown in Figure 3-34, allows you to specify the default options for creating new parts, and contains the following options:
  - **Sketch on New Part Creation.** When you develop a new part file, a sketch is automatically created on a designated plane. Inside the **Sketch on New Part Creation** area you can specify on which plane the sketch is to be created. You can select the X-Y, Y-Z, or X-Z planes. An alternative is to pick the **No new sketch** radio button if you do not want to automatically activate a new sketch whenever you create a new part.

PROFESSIONAL TIP



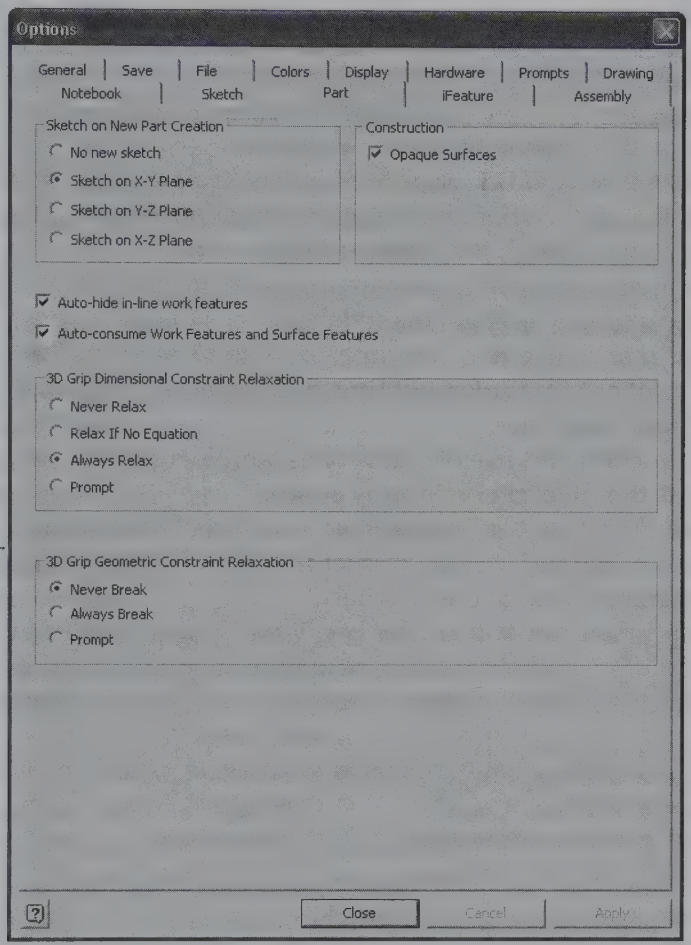
If you usually sketch a base feature profile on the same plane when beginning a model, select the plane from the **Sketch on New Part Creation** area, otherwise pick the **No new sketch** radio button to allow for greater flexibility.

  - **Parallel view on sketch creation.** Pick this check box if you want to automatically rotate the current view to a two-dimensional profile view when you begin a sketch. For example, when this check box is selected and you are currently in an isometric view, the display reorients to show the two-dimensional profile view of the new sketch plane. This option is very effective because it is typically easier to sketch on a two-dimensional flat plane, and you do not have to use view tools to reorient the model each time you sketch.

Figure 3-33.
The **Edit Dimension** dialog box.



Figure 3-34.
The **Options** dialog box, **Part** tab.



- **Auto-hide in-line work features.** Selecting this check box initiates the **Auto-hide in-line work features** tool and automatically hides consumed work features.
- **Construction.** By default, surfaces typically used for construction purposes, are displayed as translucent. However, if you want surfaces to be opaque, just as with other model geometry, pick the **Opaque Surfaces** check box in the **Construction** area.

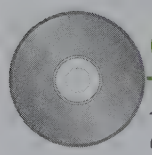
PROFESSIONAL TIP



Pick the **Rebuild All** option of the **Tools** pull-down menu to ensure that all model geometry is calculated and accounted for, especially when working with large complex models.



Complete the Exercise on the Student CD.



Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. What are iProperties and what do they represent?
2. Describe the type of information found in project properties.
3. Briefly discuss the function of status properties.
4. Explain the function and importance of Inventor view options.
5. Give the function of the zoom view option and identify two ways of accessing the option, other than through the **View** pull-down menu.
6. Discuss the function of Inventor styles, and give an example.
7. When can you activate an Inventor style?
8. Name the primary Inventor resource used for observing and modifying the characteristics of existing color, lighting, and material styles, and for creating new styles and importing styles.
9. Explain the function and use of color styles, and identify the default color.
10. Briefly discuss the function of lighting styles.
11. Discuss material styles and explain how material is intended to be a virtual representation of the real world product material. Inventor allows you to specify the type of material used to create a part.
12. Give the purpose for updating styles and identify when styles can be updated.
13. When you modify a style or create a new style, what must you do in order for the style to be effectively used in other documents, and what happens if you do not do this?
14. When purging a style, what happens to the style if it has been saved to the styles library?
15. Give the basic function of Inventor document settings.

Problems

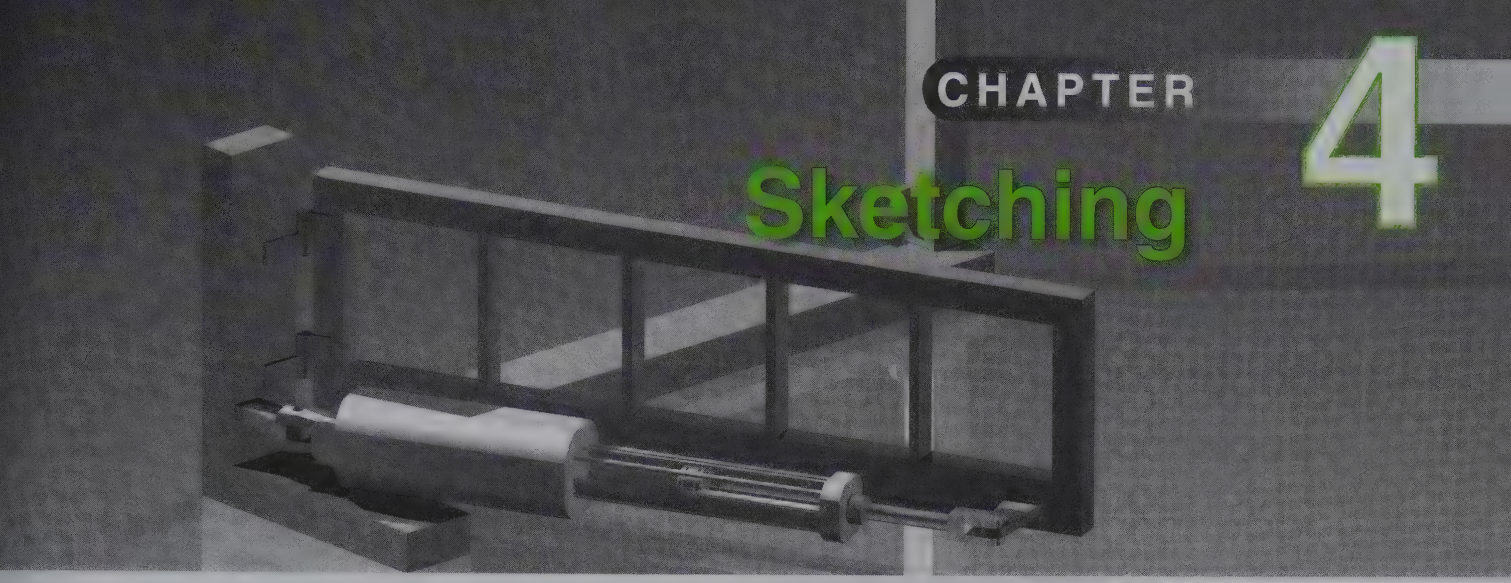
1. Launch Inventor and perform the following tasks:
   A. Open Tire Rim.ipt from the following directory: Autodesk/Inventor 11/Samples/Models/Parts/TireRim. If the Exercises and Problems project is active you should be able to locate this file quickly by picking the Samples folder in the **Locations** area of the **Open File** window.
   B. Save a copy of the tire rim part model as P3-1.ipt in your Exercises and Problems project folder.
   C. Close Tire Rim.ipt without saving and open P3-1.ipt.
   D. Access the **Physical** tab of the **Properties** dialog box, and change the material to Aluminum-6061.
   E. Use the **Color** drop-down list of the **Inventor Standard** toolbar to change the color from Black Chrome (real) to Yellow, and then to As Material.
   F. Access the **Styles and Standards Editor** and activate the Cast Steel material style. Notice the change in the graphics window.
   G. Pick the Cast Steel material style if it is not already selected, and notice the Metal-Steel color style definition.
   H. Expand the **Color** node and notice how the active color style is Metal-Steel.

I. Activate the Black color style, and notice the effect to the model.
J. Return to the Cast-Steel style of the **Materials** node and notice how the material color is still Metal-Steel.
K. Save P3-1.ipt.
L. Exit Inventor or continue working with the program.

2. Create an inch part model template by saving a copy of the Standard (in).ipt template file, found in Autodesk/Inventor 11/Templates/English/Standard (in).ipt, to your INCH Template folder. Name the copied file PART-in.ipt. Open PART-in and use the following information to prepare the custom inch template. The following specifications are required to be defined in the template. Leave all of the other items set according to the default configuration or you can modify some of the settings that are not associated with inch units. You may choose to change some of these settings once you are more familiar with Inventor. Be sure you resave your template file in Autodesk/Inventor 11/Templates/INCH Templates. This template is used in exercises and problems that require you to create a part model with inch units. Refer to Chapter 2 for more information on creating and using templates.
   A. There should not be any existing sketches or features in your template files. If the file contains a sketch, right-click on the sketch in the browser and pick the **Delete** pop-up menu option.
   B. Add the following information to the **Properties** dialog box, and leave everything else blank:
      - **Author:** Enter your name.
      - **Manager:** Enter your instructor or supervisor name.
      - **Company:** Enter your school or company name.
      - **Material:** Default
      - **Active color style:** Default
      - **Active lighting style:** Two Lights
      - **Active material style:** Default
   C. In the **Document Settings** dialog box, change the **Linear Dim Display Precision** to 0.1234, and leave all other settings according to the default configuration.
3. Create a metric part model template by saving a copy of the Standard (mm).ipt template file, found in Autodesk/Inventor 11/Templates/Metric/Standard (mm).ipt, to your METRIC Template folder. Name the copied file PART-mm.ipt. Open PART-mm.ipt and use the following information to prepare the custom metric template. The following specifications are required to be defined in the template. Leave all of the other items set according to the default configuration or you can modify some of the settings that are not associated with metric units. You may choose to change some of these settings once you are more familiar with Inventor. Be sure you resave your template file in Autodesk/Inventor 11/Templates/METRIC Templates. This template is used in exercises and problems that require you to create a part model with metric units. Refer to Chapter 2 for more information on creating and using templates.
   A. There should not be any existing sketches or features in your template files. If the file contains a sketch, right-click on the sketch in the browser and pick the **Delete** pop-up menu option.
   B. Add the following information to the **Properties** dialog box, and leave everything else blank:
      - **Author:** Enter your name.
      - **Manager:** Enter your instructor or supervisor name.
      - **Company:** Enter your school or company name.
      - **Material:** Default

C. Add the following information to the **Document Settings** dialog box, and leave everything else blank:
   - Active color style: Default
   - Active lighting style: Two Lights
   - Active material style: Default
   - Linear Dim Display Precision: 0.1234

4. Locate and open the following part from the specified path: Autodesk/Inventor 11/Samples/Model/Assemblies/Scissor/Scissor Components/blade_main.ipt. Rotate the model to an isometric view and draw a freehand sketch of the part.
5. Draw a freehand sketch of the 3D indicator. Label the X-, Y-, and Z-axes and specify the color of each. Describe the function of the 3D indicator.



CHAPTER 4

Sketching

Learning Objectives

After completing this chapter, you will be able to do the following:

- Reference and project geometry for sketches.
- Sketch lines, splines, circles, ellipses, and arcs.
- Sketch rectangles and polygons.
- Add fillets and chamfers to sketch geometry.
- Add sketch points and hole centers.
- Pattern and offset sketch objects.
- Constrain sketches using geometric constraints and dimensions.
- Create sketch text.
- Work with sketch parameters.
- Edit the geometry and constraints in sketches.
- Troubleshoot sketches using the Sketch Doctor.

Sketches usually represent the first step in the creation of a model, and are used throughout the design process. Sketches provide the foundation and profile used for developing part and assembly models that contain sketched features. This chapter introduces you to sketching and describes a number of Autodesk Inventor sketching tools. These sketching techniques are used throughout this text to create parts, assemblies, presentations, and drawings.

Sketch Concepts

Typically, the first step to creating a sketch involves sketching a basic profile using sketch tools such as **Line**, **Circle**, **Arc**, **Polygon**, and **Rectangle**. See Figure 4-1. Next, you constrain the sketch using dimensions and geometric constraints. See Figure 4-2. These are the only two steps involved in basic sketch development. However, for some sketches, you can use additional sketch tools to create more geometry and edit existing sketch data.

Consider the following concepts when developing sketches:

- It is often helpful to project the center point, located in the **Origin** folder of the browser, before you sketch any geometry. Begin your sketch from the projected

Figure 4-1.
The profile in this sketch was created with a **Two point rectangle** tool.

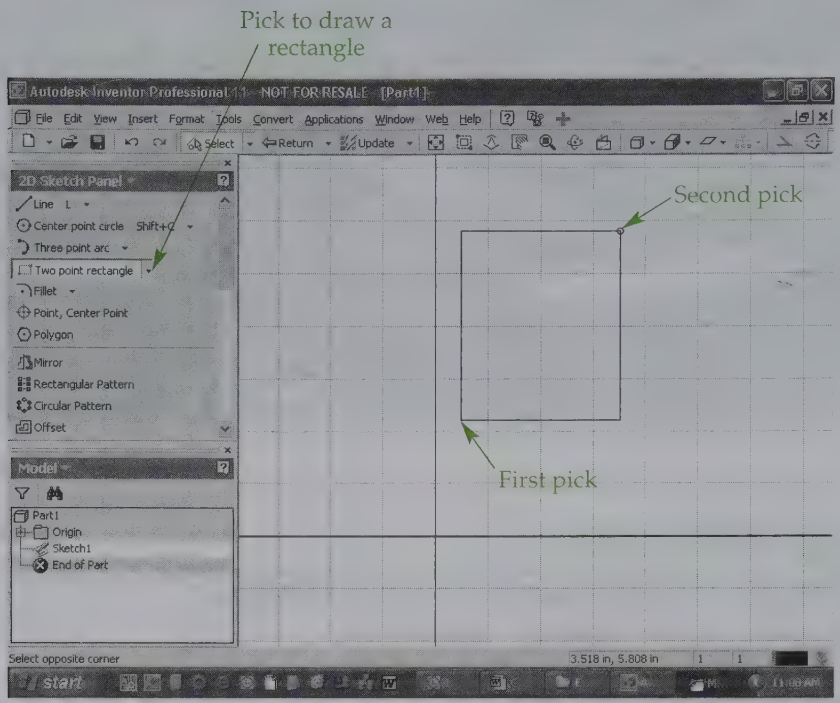
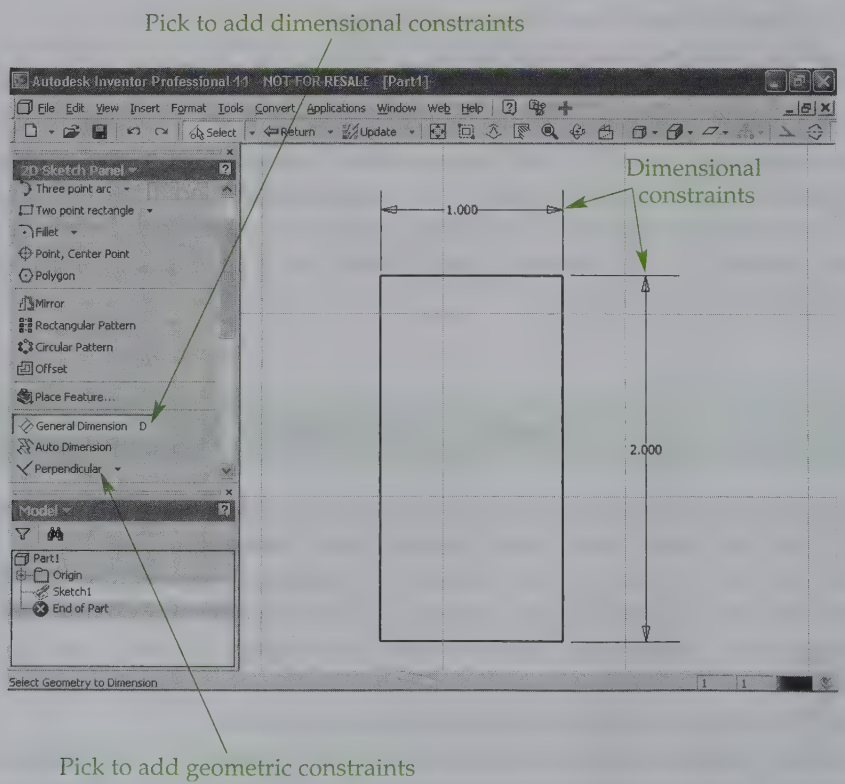


Figure 4-2.
Dimension and geometric constraints control the size and shape of the sketch.



center point in order for the sketch to become fully defined, and ensure the sketch is referenced at the 0,0,0 point of three-dimensional X,Y,Z space.

- Initial sketch geometry is often approximate and contains very few constraints. The idea is to draw the basic shape or outline of the sketch, quickly and easily, and then add dimensions and geometric constraints.
- Fully close a sketch loop when developing most profiles. If a gap or open loop exists, most features created will be volumeless surfaces.
- Develop simple, often incomplete sketch geometry, and create as many objects as possible in the feature environment. For example, use feature tools to add a chamfer, instead of drawing and dimensioning a chamfer in the sketch environment.
- Fully constrain your sketch if applicable, and do not allow the sketch to become over-constrained.
- Sketch objects according to the features, part, and drawing they help create.

Beginning a Sketch

By default, when you start a new part model, a sketch plane is automatically selected and you are ready to begin sketching. You know you are in the sketch environment when you see a sketch, Sketch1 by default, active in the browser bar and the **2D Sketch Panel** bar is displayed. If the **No new sketch** radio button of the **Options** dialog box, **Part** tab is selected, you must manually start a sketch. To begin a new sketch, select the **Sketch** button on the **Inventor Standard** toolbar and pick a plane. The sketch can also be defined by picking the plane first and then the **Sketch** button. You can do this by picking a plane from the **Origin** folder in the browser or picking an existing feature face. You can also begin a sketch by right-clicking on any face or plane, including the default work planes in the **Origin** folder, and picking the **New Sketch** pop-up menu option.

NOTE



If you choose to automatically activate a sketch by selecting one of the **Sketch on New Part Creation** radio buttons in the **Part** tab of the **Options** dialog box, you do not have to manually activate a sketch using the **Sketch** button.

Finishing a Sketch

When you are done creating a sketch and are ready to enter the feature work environment, use one of the following options to finish the sketch: right-click and select the **Finish Sketch** option from the pop-up menu, press the [S] key, pick the available feature tool from the **Part Features** panel bar or **Part Features** toolbar, or select the **Return** button on the **Inventor Standard** toolbar.

PROFESSIONAL TIP



If the default plane on which the sketch is placed or the sketch plane you select is incorrect, you must finish the sketch and then delete it. Right-click on the sketch in the browser and select the **Delete** pop-up menu option, or select the sketch in the browser and press the **Delete** key. Another option is to finish the sketch, and then redefine the sketch plane. To do this, right-click on the sketch in the browser and select the **Redefine** pop-up menu option. Now pick a different sketch plane from the **Origin** folder

Referencing and Projecting Geometry

Autodesk Inventor provides you with a few different options for using existing geometry for sketch development. Many of the reference and projection options involve using feature edges and are described in future chapters. For now, you can reference the work features available in the **Origin** folder of the browser bar. Projecting the center point, or 0,0,0 origin, onto the current sketch plane is an excellent way to begin a sketch and ensure the sketch becomes fully constrained and located at a logical point in three-dimensional space. Once the center point is projected onto the sketch plane, geometric constraints or dimensions can be used to secure the sketch to the center point. The result is a sketch that can be fully constrained and is associated with the 0,0,0 origin. Rectangle A in Figure 4-3 appears to be fully constrained, but is still actually floating in space because the sketch is not constrained to the center point. Rectangle B is constrained to the projected center point, allowing the sketch to become fully constrained.

The process of projecting work features available in the **Origin** folder is exactly the same as projecting feature edges. First, select the **Project Geometry** button of the **2D Sketch Panel** bar, **2D Sketch Panel** toolbar, or **2D Sketch Derived** toolbar. Then pick the geometry or work feature you want to reference. For example, to project the center point, access the **Project Geometry** tool and pick center point, located in the **Origin** folder of the browser.

You can automatically project the center point on a new sketch plane by picking the **Autoproject part origin on sketch create** check box from the **Sketch** tab of the **Options** dialog box. Selecting this option eliminates the need to manually project the center point each time you want to reference the origin for constraining a sketch.

NOTE



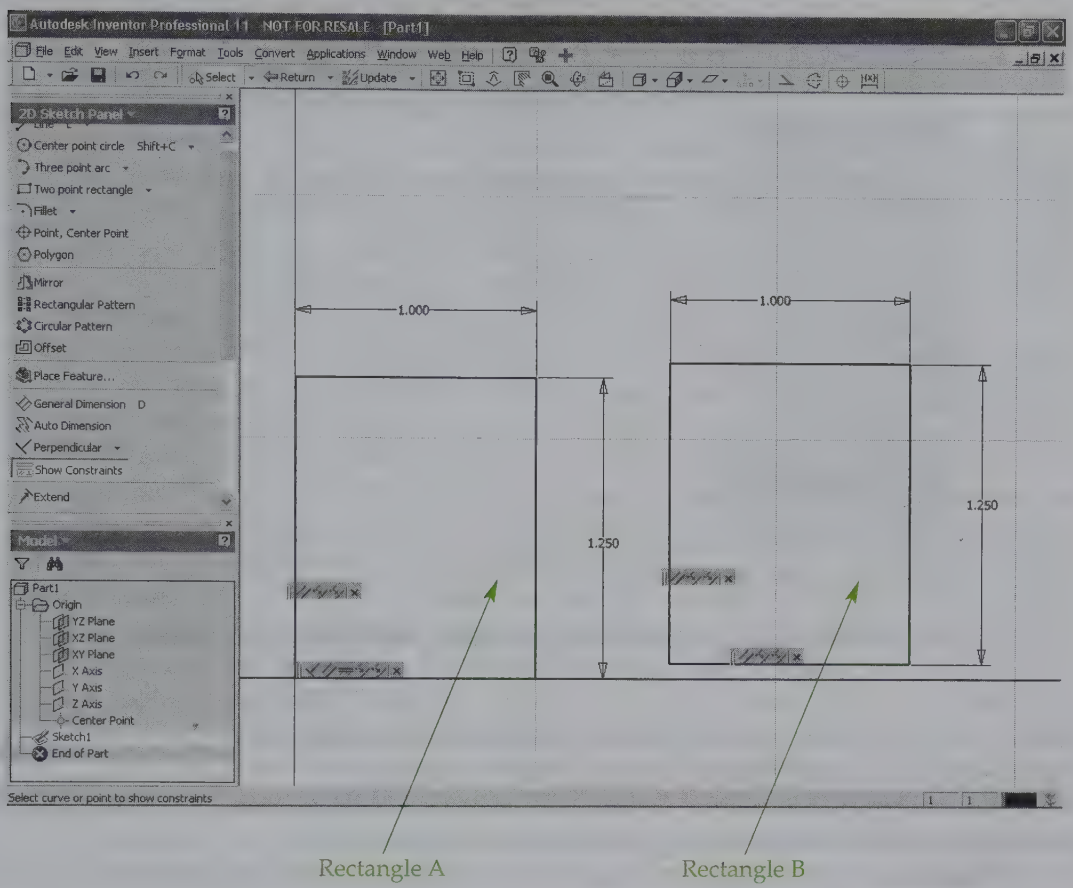
Using the **Fix** constraint tool also secures a point to a position in space. However, this technique does not fix a point to any other item such as the center point, and should be avoided.

PROFESSIONAL TIP



When creating sketch geometry, you may want to use the coordinates and measurements displayed on the right side of the status bar. These can help you draw an initial sketch and assist you in understanding geometric placement. However, the coordinates and measurements do not constrain or dimension a sketch.

Figure 4-3.
Project the center point onto the current sketch plane. Then, use dimensions and geometric constraints to constrain the sketch to the center point.



EXERCISE 4-1 Complete the Exercise on the Student CD.

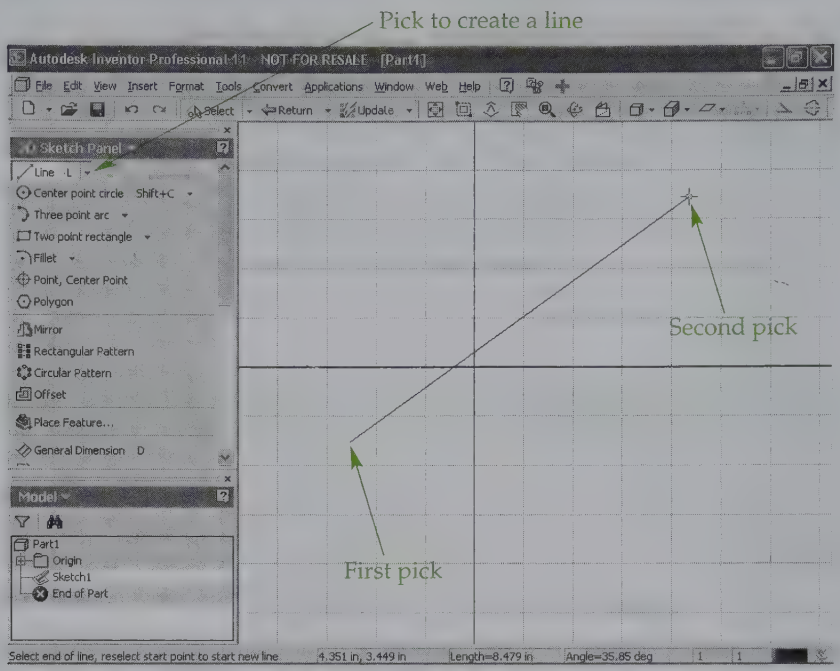
EXERCISE 4-2 Complete the Exercise on the Student CD.

Sketching Lines

To access the **Line** tool, pick the **Line** button on the **2D Sketch Panel** bar; pick the **Line** button on the **2D Sketch Panel** toolbar; press the [L] key; or pick the **Line** button on the **2D Sketch Line** toolbar.

To sketch a line, pick the point where you want to begin the line, and then pick the next point. See Figure 4-4. If you want to continue the line, pick another point. If you do not want to continue the line, press the [Esc] key, or right-click and pick the **Done** pop-up menu option. If you do not want to continue the current line, but want to draw

Figure 4-4.
To draw a line, pick the **Line** tool, a start point, and an endpoint.

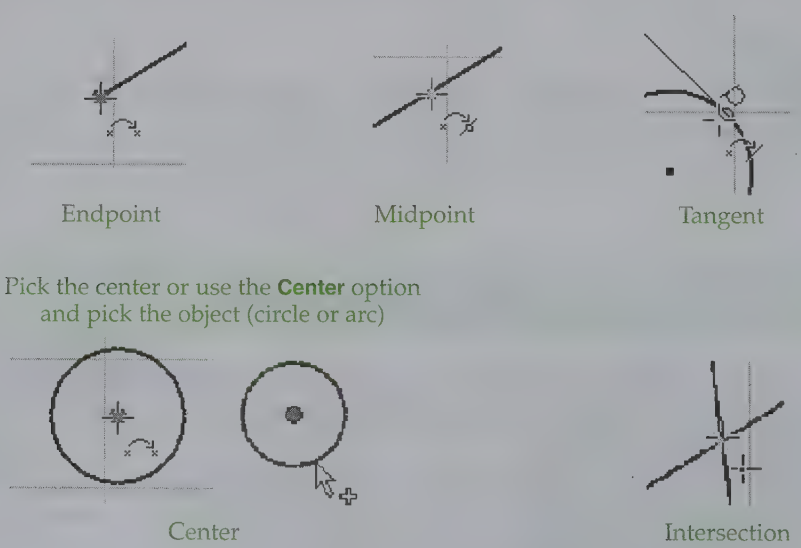


another line at another location, either close the current **Line** tool and access it again, or right-click and select **Restart** from the shortcut menu.

If you want to draw a line, or most other objects, that references existing sketch geometry, you have the following options, see Figure 4-5:

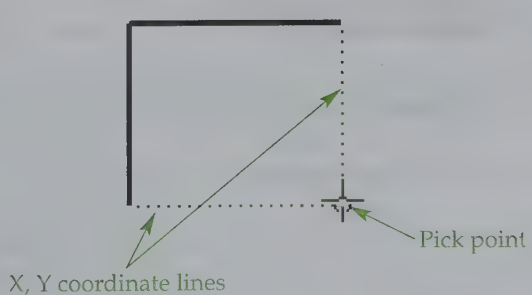
- **Endpoint.** Use this option to draw a line to or from an existing endpoint. Access the **Line** tool and move the cross and yellow dot over the apparent endpoint of an existing object. When the yellow dot turns green, gets larger, and the endpoint, or coincident, icon is visible, pick the point.

Figure 4-5.
There are many ways to reference existing objects when creating sketches.



- **Midpoint.** Use this option to draw a line to or from an existing midpoint. First, access the **Line** tool and move the cross and yellow dot over the apparent midpoint of an existing object. When the yellow dot turns green, gets larger, and the midpoint icon is visible, pick the point. You can also locate the midpoint by right-clicking, selecting **Midpoint** from the shortcut menu, and picking the object.
- **Center point.** Use this option to draw a line to or from the center of an existing circle, arc, ellipse, fillet, or other radius. First, access the **Line** tool and move the cross and yellow dot over the apparent center of an existing round object. When the yellow dot turns green, gets larger, and the coincident icon is visible, pick the point. An alternative method of locating the center point is to right-click and select the **Center** option from the shortcut menu, and pick the object, not the center point, to or from which you want to draw the line.
- **Intersection.** Use this option to draw a line to or from an existing intersection of two objects. First, access the **Line** tool and move the cross and yellow dot over the apparent intersection of the existing objects. When the yellow dot turns into a yellow cross and the intersection icon appears, pick the point. The intersection can also be located by right-clicking, selecting the **Intersection** option from the shortcut menu, and then picking the two intersecting objects to or from which you want to draw the line.
- **Tangent.** Use this option to draw an object from the tangent edge of an existing circle, arc, ellipse, fillet, or other radius. First, access the **Line** tool and move your cursor over, or near, the existing radius edge to or from which you want to draw a line. When you are close to tangent, the tangent icon appears. Pick the point and continue drawing the line or cancel the tool.
- **Coincident.** The coincident icon often appears when you move the cursor over existing geometry while using the **Line** tool. When this icon appears, the point you are about to specify exactly corresponds to a point on the existing geometry. This is the constraint that occurs when you select the projected center point, located in the **Origin** folder, when beginning a sketch.
- **X,Y Coordinates.** Use this option to draw a line from a point in space, horizontally or vertically, to a referenced point on an existing object. First, access the **Line** tool and move the cursor over, or near, the existing object to or from which you want to draw a line. Then, move the cursor away from the object, and a horizontal (X), or vertical (Y) dotted line should trail your cursor. Once you are far enough away from the existing geometry, and the dotted line is still visible, pick the point. You can also reference two points as shown in Figure 4-6.

Figure 4-6.
Use the dotted X,Y coordinate lines to align objects.



PROFESSIONAL TIP

If you do not want to use automatic referencing tools, hold down the [Ctrl] key while using the **Line** tool.

EXERCISE 4-3

Complete the Exercise on the Student CD.

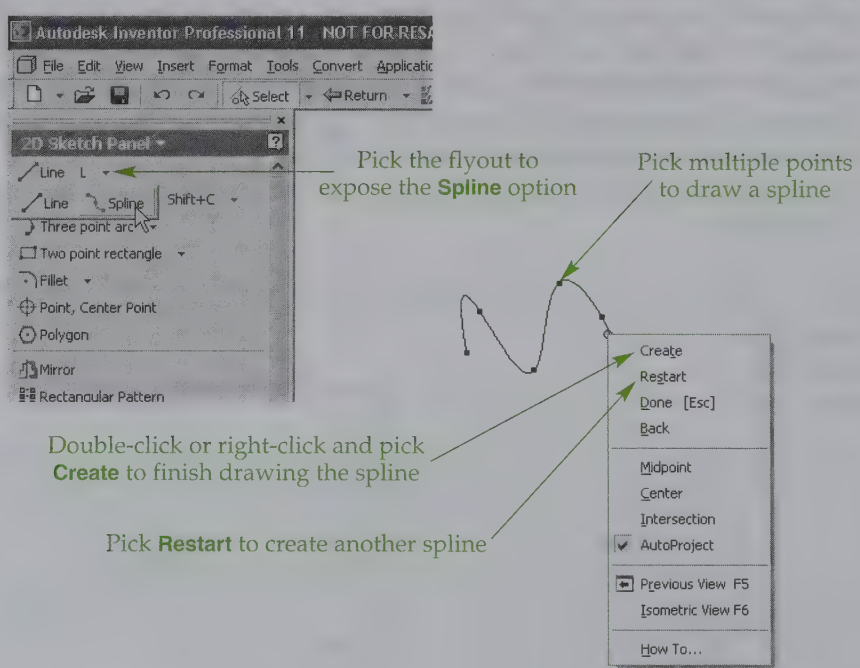
EXERCISE 4-4

Complete the Exercise on the Student CD.

Sketching Splines

Pick on the flyout arrow next to the **Line** tool of the **2D Sketch Panel** bar to reveal the **Spline** tool. See Figure 4-7. A *spline* is similar to a line, but creates a single line with rounded corners instead of multiple lines with angled corners. To draw a spline, pick the point where you want to start and continue to pick points. You can double-click the last point or right-click and pick **Create** to end the spline.

Figure 4-7.
A spline is a single object made of smooth rounded corners.



NOTE



Though you may pick several times to create a spline, only one object is created. Picking more than two times while using the **Line** tool creates more than one object.

If you want to sketch another spline, either close the current **Spline** tool and access the **Spline** tool again, or right-click and select **Restart** from the shortcut menu. Then repeat the process as previously described. If you do not want to continue the **Spline** tool, press the [Esc] key, right-click, and pick the **Done** option, or pick another tool.

NOTE



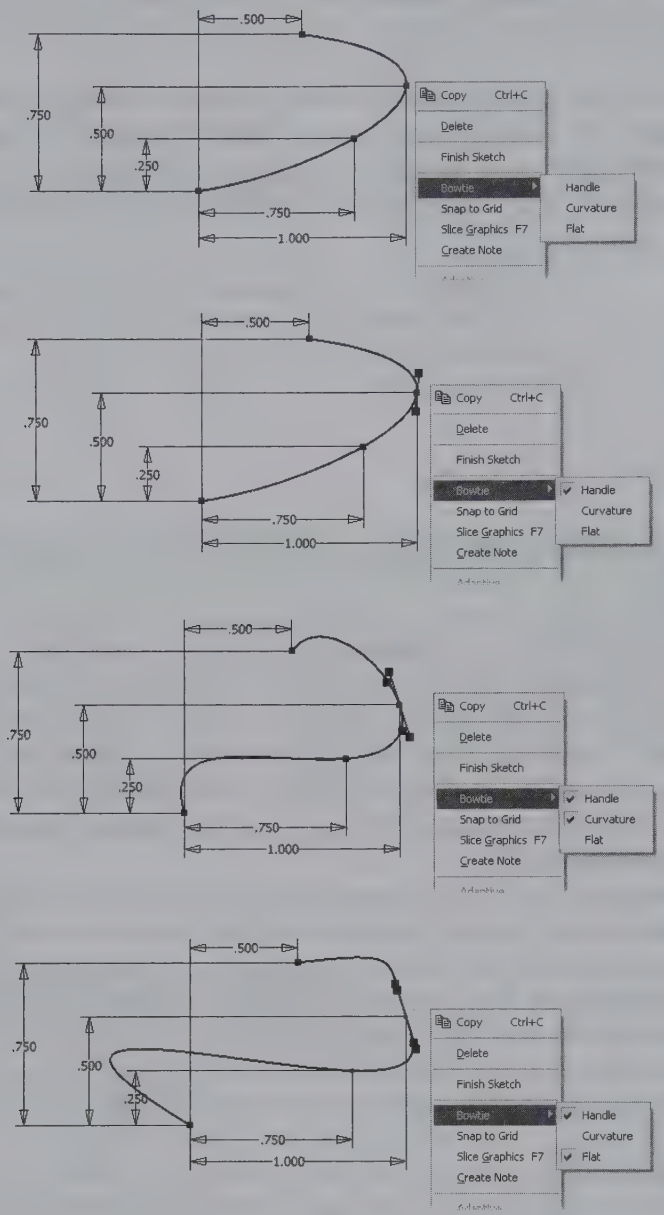
Each of the techniques used to reference existing geometry with the **Line** tool apply when sketching splines.

Editing Splines

After creating a spline, right-click on it to view the following options to edit the spline:

- **Bowtie.** This option contains a cascading submenu with three tools that allow you to further adjust the spline shape. You can pick one, two, or all three selections depending on the application. To better understand the concept of spline bowties, see Figure 4-8, which shows a spline that appears to be constrained. The curves that define a spline can be manipulated and constrained in a number of ways using bowtie handlebars. To add a bowtie, right-click on a spline curve or point, and then choose either **Handle**, **Curvature**, or **Flat** from the cascading submenu.
  Use the **Handle** option to add a handlebar. You can drag handlebar endpoints, or *grips*, to the desired location, or constrain the handlebar using dimensions and geometric constraints. Choose the **Curvature** option to add both a handlebar and a curvature bar. The curvature bar adds control to the **Handle** option by allowing you to dimension the spline curve radius. Finally, when you pick the **Flat** option, all three bowtie tools become active, adding a handlebar and a flat curvature bar. The flat option removes the radius from the specified spline curve.
- **Fit Method.** Pick this option to access a cascading submenu with three menu options. The options are used to define how spline points are positioned in reference to the spline. The **Standard** option is active by default and displays smooth curves between spline points. Pick the **Minimum Energy** option to create a spline with smoother and more even curves than is provided with the **Standard** fit, or select the **AutoCAD** option to define AutoCAD's spline fit technique.
- **Insert Point.** Pick this option to add another control point to the spline. Pick a point on an existing spline to insert the additional spline joint.
- **Close Spline.** Select this option to change an open spline into one where the beginning and end points are connected.
- **Display Curvature.** When you choose this menu option, curvature lines, or *combs*, are added to the spline to help illustrate and analyze the curvature of the spline. If you select a different fit method from the **Fit Method** cascading submenu, you may also want to pick the **Display Curvature** option to help

Figure 4-8.
Notice the effects of the bowtie options on splines created with the same dimensions.



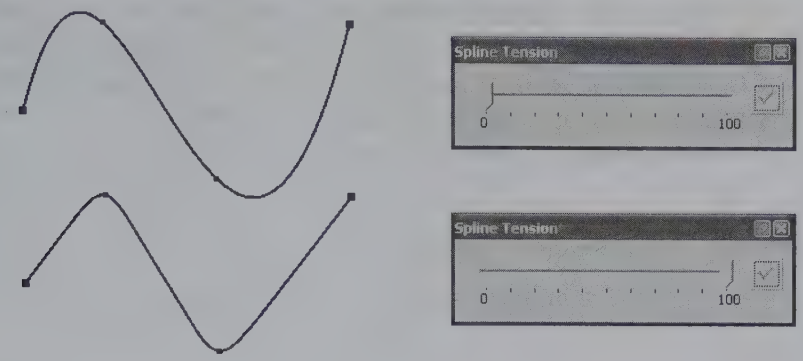
visualize the fit. Once you select **Display Curvature**, the **Setup Curvature Display** option becomes available from the pop-up menu when you right-click on a spline. Pick **Setup Curvature Display** to access the **Curvature Settings** dialog box. The **Curvature Settings** dialog box contains slide bars that are used to increase or decrease comb density and scale.

NOTE



The **Display Curvature** and **Setup Curvature Display** options are available when you right click on any curve, not just a spline.

Figure 4-9.
These splines were identical until the tension was increased.



- **Spline Tension.** This option allows you to adjust the amount of arc, or arc radius, between spline points. When you select **Spline Tension**, the **Spline Tension** dialog box is displayed. Use the slide bar to adjust the spline curve tension from 0 to 100, and then pick the check mark button. See Figure 4-9. If you specify a spline fit method other than minimum energy and then increase spline tension, the fit method is automatically converted to minimum energy.

NOTE



Many of the spline bowtie, fit, curvature, and tension adjustments are unnecessary for basic spline sketching, but can be used to analyze and fully control the display of a spline when needed.

PROFESSIONAL TIP



When creating splines with multiple segments, you can remove the latest segment and return to the previous spline point by right-clicking and selecting the **Back** option from the shortcut menu, or by pressing the [Backspace] key.



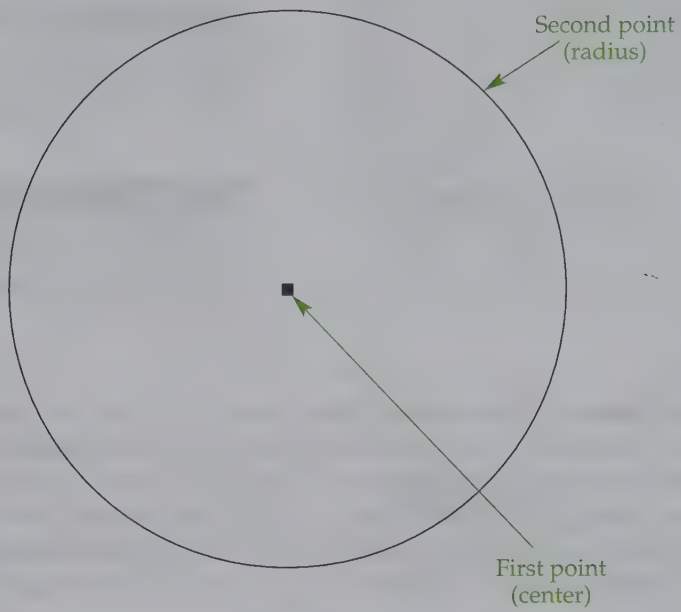
Complete the Exercise on the Student CD.

Sketching Circles and Ellipses

Circle and ellipse tools are found in a flyout button that is initially set as the **Center point circle** tool. To access the **Center point circle** flyout button pick the **Center point circle** button on the **2D Sketch Panel** bar, pick the **Center point circle** button on the **2D Sketch Panel** toolbar, press [Ctrl] + [C].

To draw a center point circle, pick the point you want to designate as the center of the circle, drag the circle edge away from the center point, and pick the location to define circle edge. See Figure 4-10. If you want to draw another circle, pick a new center

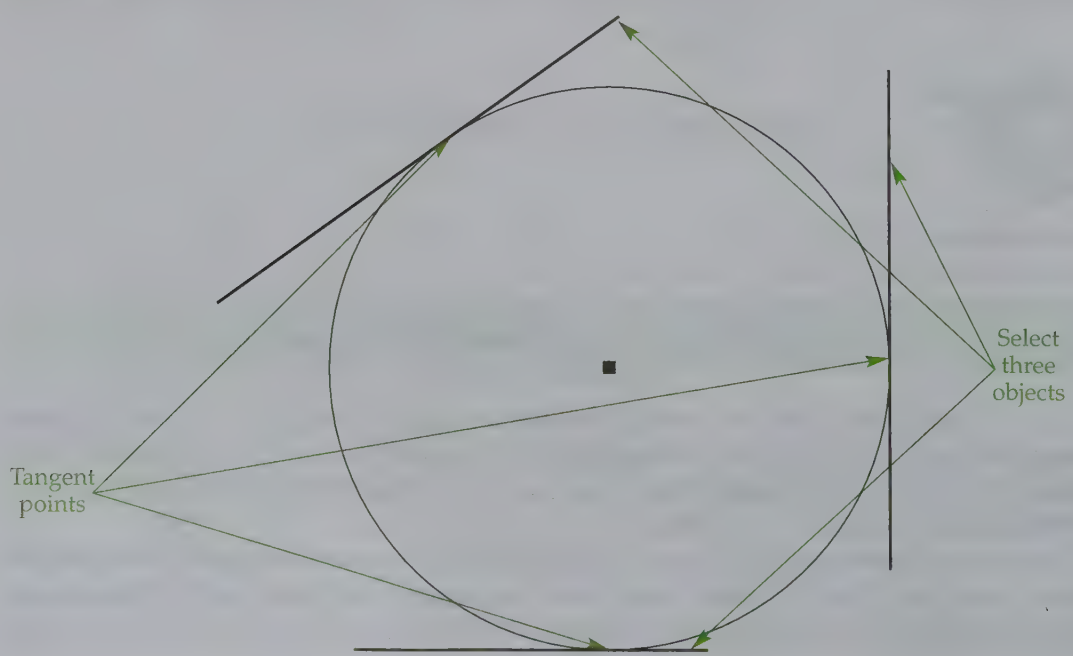
Figure 4-10.
When creating a circle, the first pick locates the center and the second pick defines the radius.



point. If you do not want to draw another circle, press the [Esc] key, right-click and pick the **Done** option, or pick another tool.

The default **Center point circle** tool is a flyout button that also contains options for drawing a tangent circle and an ellipse. To draw a tangent circle, you must have three reference objects available, such as lines, to define the circle. If these objects are present, access the **Tangent circle** tool from the circle flyout button and pick the three existing reference objects. A circle is automatically placed between the objects as shown in Figure 4-11. If you want to draw another tangent circle, pick a new group of objects.

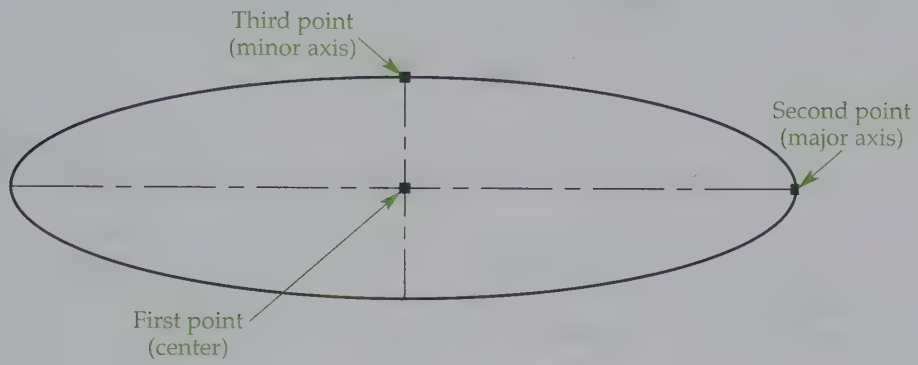
Figure 4-11.
The **Tangent circle** tool is used to draw a circle tangent to three lines.



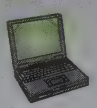
If you do not want to draw another circle, press the [Esc] key, right-click and pick the **Done** option, or pick another tool.

An ***ellipse*** is similar to a circle, but contains a major and a minor axis. To draw an ellipse, access the **Ellipse** tool from the circle flyout button. Then, specify the center point, followed by choosing the first axis endpoint, and then the second axis endpoint. See Figure 4-12. If you want to draw another ellipse, pick a new center point. If you do not want to draw another ellipse, press the [Esc] key, right-click and pick the **Done** option, or select another tool.

Figure 4-12.
An ellipse is defined by the center point, major axis, and minor axis.



PROFESSIONAL TIP

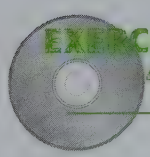


Each of the techniques used to reference existing geometry with the **Line** tool applies when sketching using the **Center point circle**, **Tangent circle**, and **Ellipse** tools. Use these referencing options to select the center point and outside radius/endpoint of radii and axes.

NOTE



If you begin to draw a circle or ellipse at an incorrect location, right-click and select the **Restart** option from the shortcut menu while the tool is still active. Then, pick where you want to replace the first point.



EXERCISE 4-6

Complete the Exercise on the Student CD.

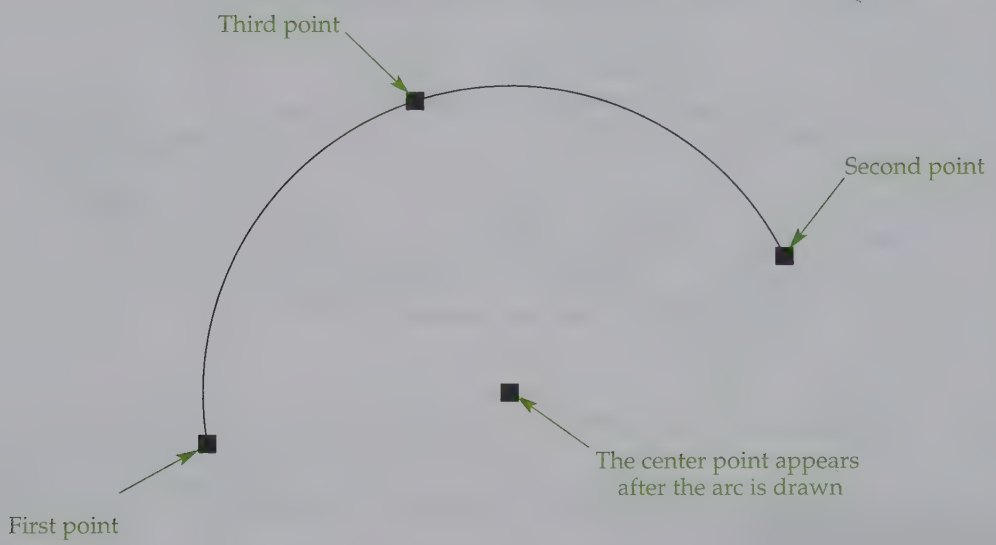
Sketching Arcs

Arcs tools are found in a flyout button initially set as the **Three point arc**. To access the **Three point arc** tool pick the **Three point arc** button on the **2D Sketch Panel** bar; pick

the **Three point arc** button on the **2D Sketch Panel** toolbar; or pick the **Three point arc** button on the **2D Sketch Arc** toolbar.

To draw a three-point arc, pick the first point of the arc, followed by selecting the endpoint, and finally pick the arc radius. See Figure 4-13. If you want to draw another three point arc, pick a new point, or press the [Esc] key, right-click and pick the **Done** option, or pick another tool.

Figure 4-13.
When drawing a three-point arc, the first and second points define the ends and the third point defines the size and direction of the arc.



Picking the arc flyout button allows you to access the other two arc tools. See Figure 4-14. One of the options is the **Center point arc** tool. To create a center point arc, first pick the center point of the arc radius. See Figure 4-15. Then, drag the dashed line out and specify the radius and the arc start point. Finally, drag the centerline around and specify the arc endpoint. Repeat the steps to draw another center point arc, or press the [Esc] key, right-click and pick the **Done** option, or pick another tool.

The third method for drawing an arc is to use the **Tangent arc** tool. To draw a tangent arc, you must have a reference object available, such as a line, to define the arc. See Figure 4-16. If an object is present, access the **Tangent arc** tool and pick the existing reference object. An arc automatically begins at the object. If you want to draw another tangent arc, pick new geometry, or press the [Esc] key, right-click and pick the **Done** option, or pick another tool.

Figure 4-14.
Pick the flyout next to the **Three point arc** tool to reveal the other arc creation tools.

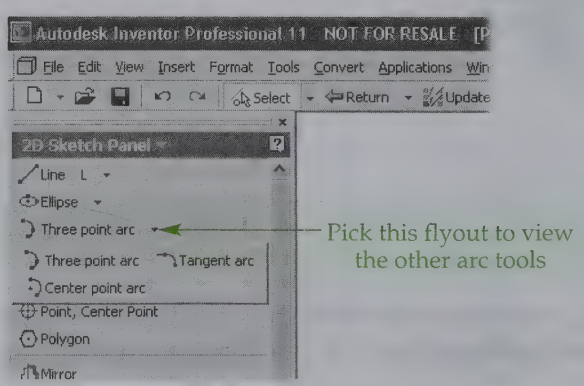


Figure 4-15.
Drawing an arc using the **Center Point Arc** tool.

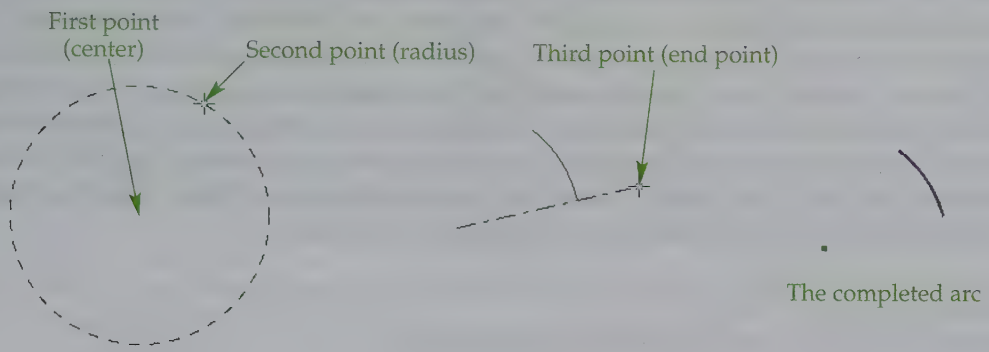
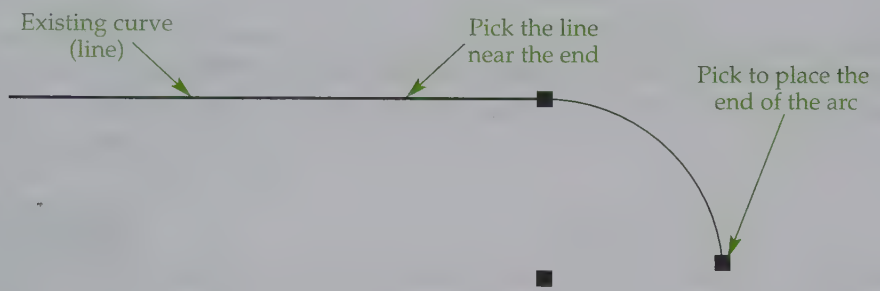
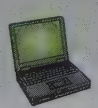


Figure 4-16.
Use the **Tangent arc** tool to create an arc that is tangent to an existing object.



PROFESSIONAL TIP



Each of the methods used to reference existing geometry with the **Line** tool apply when using the **Three point arc**, **Center point arc**, and **Tangent arc** tools. Use these referencing options to select the center point and outside radii/endpoint of your radius and axes.

NOTE



If you begin to draw an arc at an incorrect location, right-click and select the **Restart** option from the shortcut menu while the tool is still active. Then pick where you want to replace the first point.



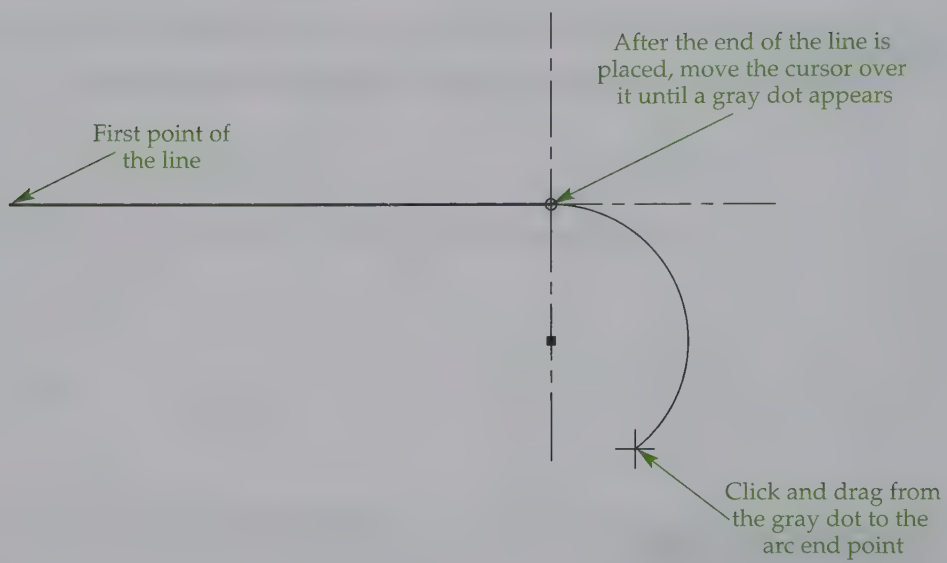
EXERCISE 4-7

Complete the Exercise on the Student CD.

Sketching Arcs Using the Line Tool

Autodesk Inventor contains a very useful tool that allows you to create arcs while using the **Line** tool. To create an arc with this method, pick the **Line** tool and create one or more lines as needed, but do not exit the **Line** tool. While still in the **Line** tool, move the cursor over the last specified point until the dot turns gray. Now, hold down the left mouse button, move the arc endpoint to its location, and release the left mouse button. See Figure 4-17. If you want to draw another arc, use the same method by moving your cursor over the arc endpoint. You can also continue to draw a straight line from the arc. If you do not want to draw another arc or line, press the [Esc] key, right-click and pick the **Done** option, or pick another tool.

Figure 4-17.
Inventor allows you to create an arc using the line tool.





Complete the Exercise on the Student CD.

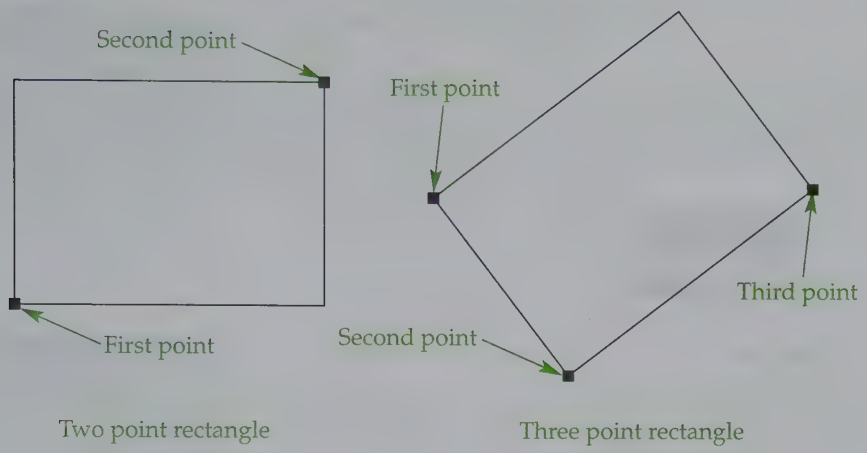
Sketching Rectangles Using Rectangle Tools

Two Autodesk Inventor rectangle tools are found in a flyout button that is initially set as the **Two point rectangle** tool. To access the **Two point rectangle** tool, pick the **Two point rectangle** button on the **2D Sketch Panel** bar; pick the **Two point rectangle** button on the **2D Sketch Panel** toolbar; or pick the **Two point rectangle** button on the **2D Sketch Rectangle** toolbar.

To draw a two point rectangle, pick the first corner of the rectangle, followed by the opposite, diagonal corner. See Figure 4-18. If you want to draw another two-point rectangle, pick a new point, or press the [Esc] key, right-click and pick the **Done** option, or select another tool.

Picking the rectangle flyout button allows you to access the **Three point rectangle** tool. To create a rectangle using the **Three point rectangle** tool, pick the first rectangle

Figure 4-18.
Use the **Two point rectangle** tool to draw a horizontally constrained rectangle. Use the **Three point rectangle** tool to draw a rectangle at an angle.



corner, drag the line and cursor out, and specify the second rectangle corner. Finally, drag the rectangle out and specify a third corner. If you want to draw another three-point rectangle, pick new points, or press the [Esc] key, right-click and pick the **Done** option, or select another tool.

PROFESSIONAL TIP



You can create a rectangle using the **Line** tool. However, usually it is much easier to use one of the rectangle options, because it only requires two or three picks, and the parallel and perpendicular constraints are already added.

NOTE



If you begin to draw a rectangle at an incorrect location, right-click and select the **Restart** option from the shortcut menu while the tool is still active. Then pick where you want to replace the first point.

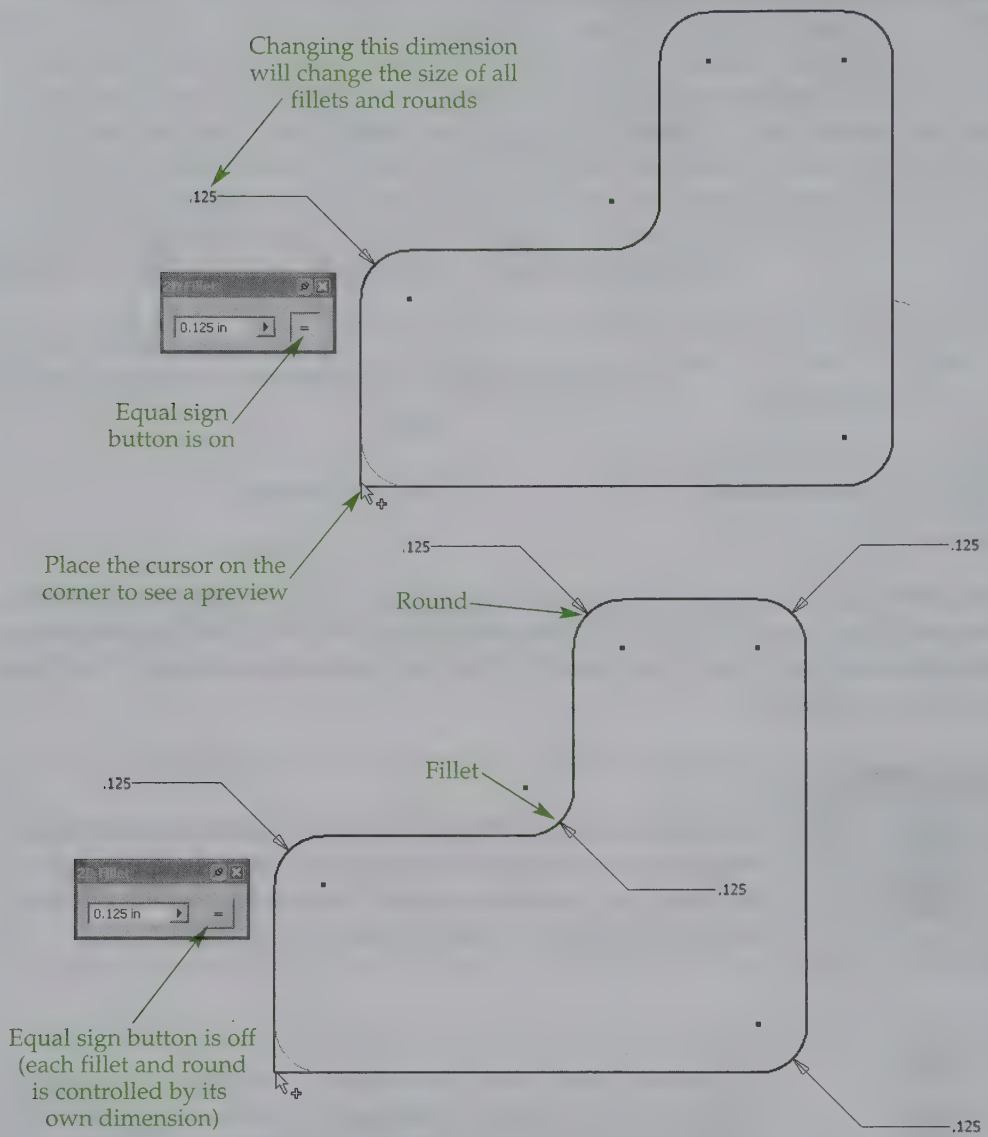


Complete the Exercise on the Student CD.

Adding Sketch Fillets, Rounds, and Chamfers

The **Fillet** tool is used to add fillets and rounds—*fillets* are rounded interior corners and *rounds* are rounded exterior corners. To access the **Fillet** tool, pick the **Fillet** button on the **2D Sketch Panel** bar or **2D Sketch Panel** toolbar. Selecting the **Fillet** button opens the **2D Fillet** dialog box, shown in Figure 4-19. To create a fillet or round, first specify a radius in the edit box by typing or picking a value from the list. Then, select the **Equal**

Figure 4-19.
When using the **Fillet** tool, pick two lines or their intersection to add a fillet or round.



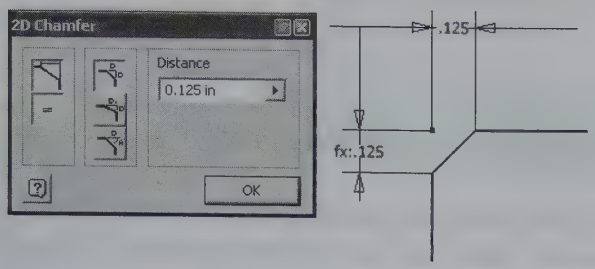
button if you want to create multiple fillets with equal radii, without adding additional dimensions. Next, you can see a preview of the fillet or round by moving your cursor to the corner of the lines or by selecting one line and moving your cursor over the next line. If the fillet or round looks acceptable in the preview, pick the appropriate lines.

NOTE

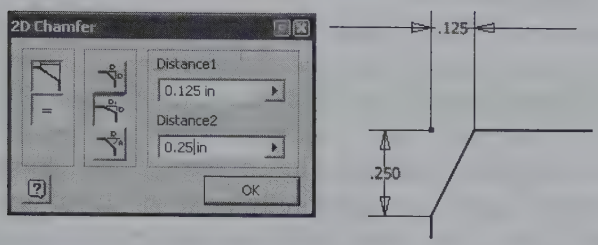
You can also create a fillet or round by picking the **Fillet** tool after selecting a corner or holding down the [Shift] key while picking two lines.

To access the **Chamfer** tool, pick the **Chamfer** button from the **Fillet/Chamfer** flyout button on the **2D Sketch Panel** bar or **2D Sketch Panel** toolbar. Selecting the **Chamfer** button opens the **2D Chamfer** dialog box, shown in Figure 4-20. To create a chamfer, access the **Chamfer** tool, and specify options in the **2D Chamfer** dialog box. Next, you can see a preview of the chamfer by moving your cursor to the corner of the lines, or by selecting one line and moving your cursor over the next line. If the chamfer looks acceptable in the preview, select the corner or the two intersecting lines where you want the chamfer. For unequal-distance

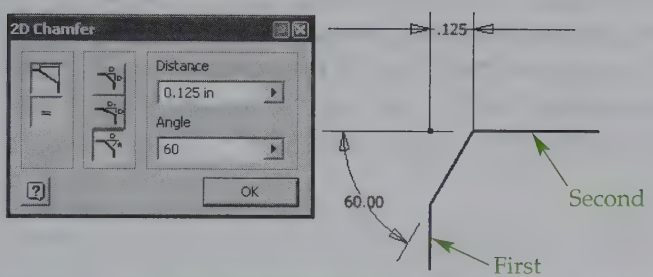
Figure 4-20.
The options in the **2D Chamfer** dialog box control the dimensioning of chamfers.



Equal distance
The fx: .125 dimension is always equal to the .125 dimension



Unequal distance
Each distance can be changed without affecting the other



Distance and angle
The distance dimension is between the intersection and the first selected line. The angle is measured from the second selected line



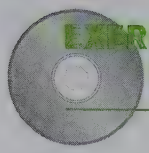
Create dimension
Dimensions are automatically added to chamfers only when this button is on

chamfers, the first line selected corresponds to Distance2, and the second line selected corresponds to Distance1. Similarly, for distance and angle chamfers, the first line selected corresponds to the **Angle**, and the second line selected corresponds to the **Distance**.

PROFESSIONAL TIP



Typically, you do not want to apply geometry such as fillets, rounds, and chamfers to a sketch, because it is usually easier to place a fillet or chamfer on a feature rather than a sketch. In addition, complicated sketch geometry that includes objects such as fillets, rounds, and chamfers can become difficult to deal with as you progress through different design stages.



Complete the Exercise on the Student CD.

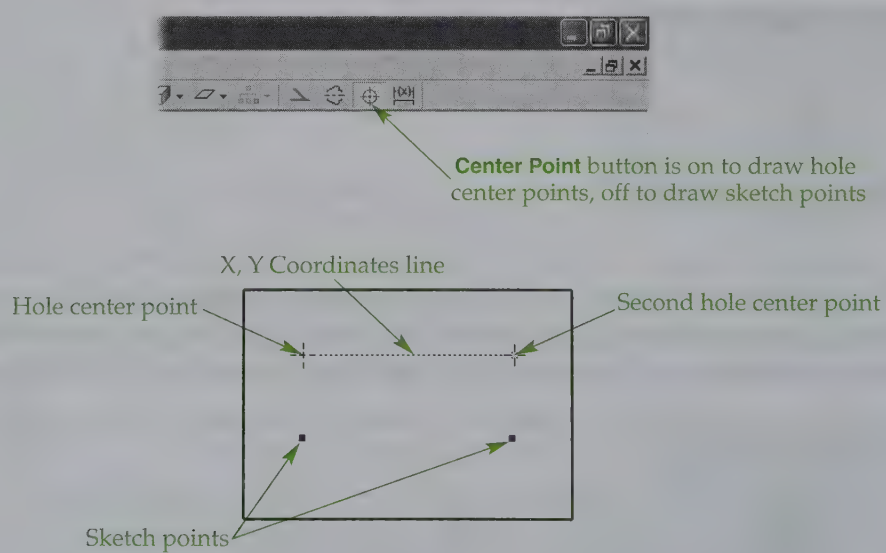
Adding Sketch Points and Hole Centers

Use the **Point,** tool to create sketch points or to place a hole center point. ***Sketch points*** are used for construction purposes and help you develop sketch geometry. Center points are used to define the location of center points for features that reference center points, such as holes and sheet metal punches.

The default **Point,** tool style is a hole center. A hole center should reference some type of feature. As a result, hole centers are typically placed on feature faces. To place a hole center, pick the **Point,** button on the **2D Sketch Panel** bar or **2D Sketch Panel** toolbar. Then pick the location where you want to place the hole center point. See Figure 4-21. If you want to add another hole center, pick a new point. The dotted line of the **X,Y Coordinates** automatic constraint indicates the new hole center is horizontally or vertically aligned with another hole center or sketch point. If you do not want to draw another hole center, press the [Esc] key, right-click and pick **Done**, or choose another tool.

In order to change the default **Point,** tool style from a hole center to a sketch point, you must deselect the button of the **Inventor Standard** toolbar. To create a sketch point, either deselect the button and then access the **Point,** tool, or pick an existing hole center and deselect the button. Sketch points are placed in a sketch exactly the same as a hole center, but are represented as dots rather than the hole center line.

Figure 4-21.
Place hole center points and sketch points in a sketch to locate features that will be added later.



NOTE

Each of the techniques used to reference existing geometry with the **Line** tool also apply when sketching with the **Point, Center Point** tool. Use these referencing options to place a hole center or sketch point in reference to existing geometry.

EXERCISE 4-11

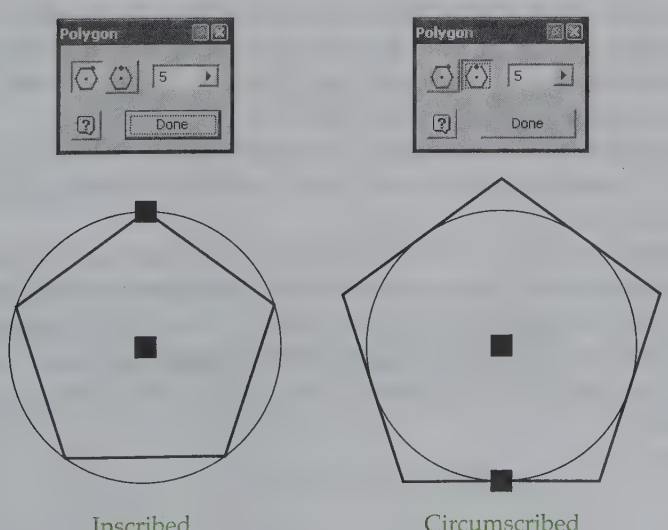
Complete the Exercise on the Student CD.

Sketching Regular Polygons Using the Polygon Tool

A *polygon* is a geometric shape with three or more sides, such as a triangle, square, or hexagon. A *regular polygon* has sides that are equal in length and symmetrical about a common center. See Figure 4-22. The **Polygon** tool is very useful for sketching regular polygons, which is a shape with three or more equal length sides and equal angles between sides. To access the **Polygon** tool pick the **Polygon** button on the **2D Sketch Panel** bar or the **2D Sketch Panel** toolbar.

Selecting the **Polygon** tool opens the **Polygon** dialog box. Specify how you want to create the polygon by selecting either the **Inscribed** or **Circumscribed** button. An *inscribed* polygon is measured from the polygon corners, while a *circumscribed* polygon is measured from the polygon flats. Next, define the type of polygon you want to sketch by entering or selecting the number of sides using the **Number of sides** edit box. Now, pick the center point of the polygon. Finally, select the edge or corner of the shape. If you chose the **Inscribed** button, the second point you pick defines the distance from the center of the polygon to the polygon corners. If you chose the **Circumscribed** button, the second point you pick defines the distance from the center of the polygon to

Figure 4-22.
The number in the edit box determines the number of sides in the polygon. The circles are added to illustrate the difference between inscribed and circumscribed.



the polygon flats. After you sketch a polygon, the **Polygon** tool remains open allowing you to continue creating polygons. To exit, press the [Esc] key, right-click and pick the **Done** option, choose the **Done** button on the **Polygon** dialog box, or select another tool.

NOTE



Each of the techniques used to reference existing geometry with the **Line** tool apply when sketching using the **Polygon** tool. Use these referencing options to select the center point and polygon corner or edge.



EXERCISE 4-12 Complete the Exercise on the Student CD.

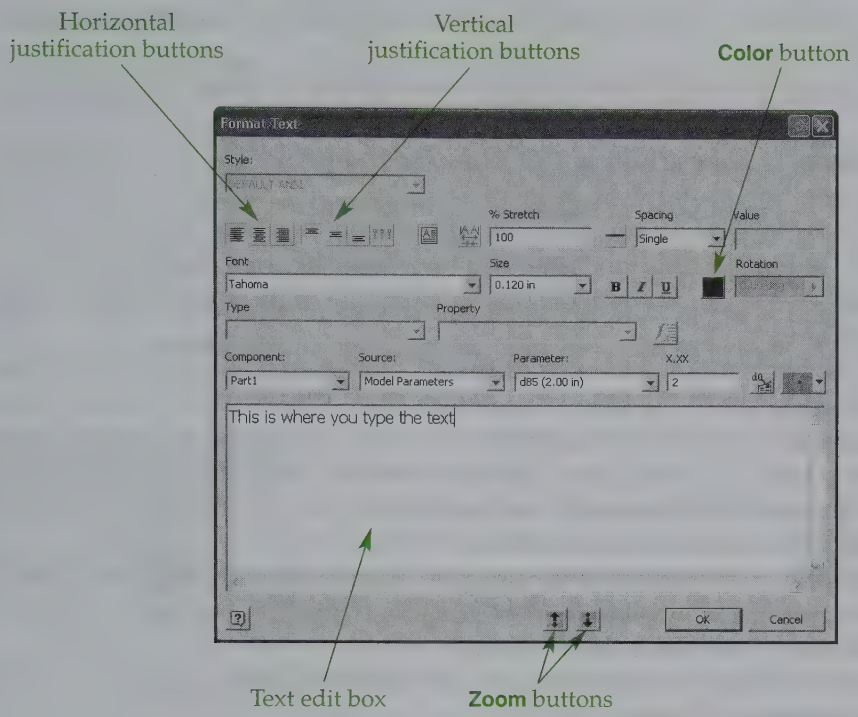
Adding Sketch Text

Several model and drawing applications require that you add text to a sketch. In models, sketch text can be used for creating features such as embossments and extrusions. In drawings, sketch text is used in a variety of functions including creating or editing title blocks, revision blocks, and notes. The following information focuses on adding sketch text to a model.

Text is added to a sketch using the **Text** tool, accessed by picking the **Create Text** button on the **2D Sketch Panel** bar or **2D Sketch Panel** toolbar. Once you access the **Text** tool, use the cursor to select a point or hold down the left mouse button and drag the cursor to create a window in the graphics window. The point you select, or window you create, defines the location and extents of the text, and the **Format Text** dialog box is displayed. See Figure 4-23. Use the following options in the **Format Text** dialog box to define the type of text you want to sketch. Only model sketch text is described:

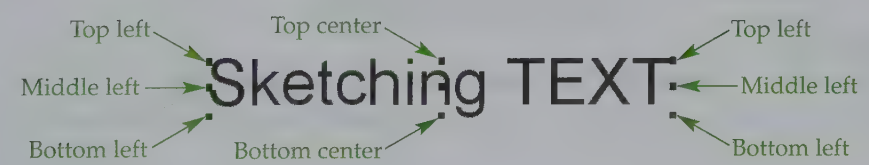
- **Justification.** These buttons allow you to control the left, center, right, top, middle, and bottom justification, or placement, of the text.
- **Text Box.** Selecting this button adds a construction box around your text. You can use the text box just as you would other construction geometry, including adding dimensions and geometric constraints to define the size and location of your text. You can also turn the text box function on or off, once the text has been created, by right-clicking and selecting or deselecting the **Text Box** pop-up menu option. If you do not use a text box, a single sketch point is placed in reference to the selected justification. The point can be located using dimensions and geometric constraints.
- **% Stretch.** This edit box allows you to define the amount of stretch, or width, of the text. To create a narrower text, enter a value less than 100, and greater than 100 for wider text.
- **Line Spacing.** This drop-down list allows you to specify the distance between multiple lines of text. When you select the **Multiple** option, the **Value** edit box becomes useable, allowing you to define a smaller or greater spacing ratio, such as triple or half. When you select the **Exactly** option, the **Value** edit box becomes available, allowing you to enter a specific distance between multiple lines of text.

Figure 4-23.
The **Format Text** dialog box appears after picking a location for new text and editing existing text.



- **Single Line Text.** Pick this button if you only want to create a single line of text. When you pick the **Single Line Text** button, the following options become available:
  - **Fit Text.** Pick this button if you want to fit your text into a specific space, modifying the text width as needed. When you select the **Fit Text** button the text width, or % stretch, varies depending on the size of the area you pick.
  - **Baseline Justification.** Text that is baseline justified has no vertical justification. The text can only be justified as left, center, or right. See **Figure 4-24**.
- **Font.** Use this drop-down list to define the font style for the selected text.

Figure 4-24.
Standard text and baseline text have different justification options.



Standard text justification options



Baseline text justification options

- **Size.** This drop-down list allows you to choose or enter a value for the size, or height, of the selected text.
- **Format.** These buttons define text display options. Pick the **Bold** button to create bold text, the **Italic** button to create italicized text, and the **Underlined** button to underline text.
- **Color.** Pick this button to display the **Color** dialog box and define the text color.
- **Rotation Angle.** Use these buttons to define the rotation of the text. A 0° rotation generates text horizontally from left to right, a 270° rotation generates text vertically from top to bottom, a 180° rotation generates text horizontally from right to left, and a 90° rotation generates text vertically from bottom to top.
- **Component.** This drop-down list contains the name of the current file in which you are working.
- **Source.** This drop-down list specifies the type or source of parameters to use. ***Model parameters*** are specific parameters that relate to the model. These parameters are added when you insert a model view or add additional model information, such as dimensions. ***User parameters*** are additional parameters added to the model. If you do not create any user parameters, the **User Parameters** source option is not available.
- **Parameter.** This drop-down list contains all the parameters available in the selected source. The parameters correspond to the parametric model or parameters used to define model geometry.
- **X.XX.** Use this edit box to define the precision of the displayed value by entering the number of decimal places.
- **Add Parameter.** Once you select a parameter and fully define parameter options, pick this button to add the parameter to the **Text** edit box.

You can also add a symbol to your text by selecting a symbol from the **Symbols** flyout button. If needed, you may want to zoom in or out on the text in the **Text** edit box by picking the **Zoom In** or **Zoom Out** buttons.

NOTE



Zooming in or out on the text in the **Text** edit box does not change the physical size of the text or symbols.

Once you choose the text characteristics and enter the desired text in the **Text** edit box, pick the **OK** button to exit the **Format Text** dialog box. Continue placing text, or exit the **Text** tool by right-clicking and selecting the **Done** menu option, pressing the [Esc] key, or accessing a different tool.



Complete the Exercise on the Student CD.

Patterning Sketch Geometry

Autodesk Inventor provides you with three tools that allow you to pattern sketch geometry. A ***sketch pattern*** is created anytime you place multiple arranged copies, or patterns, of sketches. Although typically features, not sketches, are patterned, you may find it helpful to pattern sketched shapes for some applications. Sketched objects can be

mirrored using the **Mirror** tool, patterned in a rectangular fashion using the **Rectangular Pattern** tool, and patterned in a circular fashion using the **Circular Pattern** tool.

Mirroring Sketch Objects

The sketch **Mirror** tool allows you to create a mirrored copy of sketch geometry across a line of symmetry. In order to mirror an object, you must first sketch a line of symmetry, if an existing line is not available. See Figure 4-25. Then, access the **Mirror** dialog box by picking the **Mirror** button on the **2D Sketch Panel** bar or **2D Sketch Panel** toolbar. By default, the **Select** button should be active, allowing you to select all the objects you want to mirror. Once you choose each of the items, pick the **Mirror line** button, and select the line of symmetry. Finally, pick the **Apply** button to create the mirrored copy. Continue mirroring features as required, or exit the **Mirror** dialog box by selecting the **Done** button, pressing the [Esc] key, right-clicking and picking the **Done** option, or accessing another tool.

PROFESSIONAL TIP

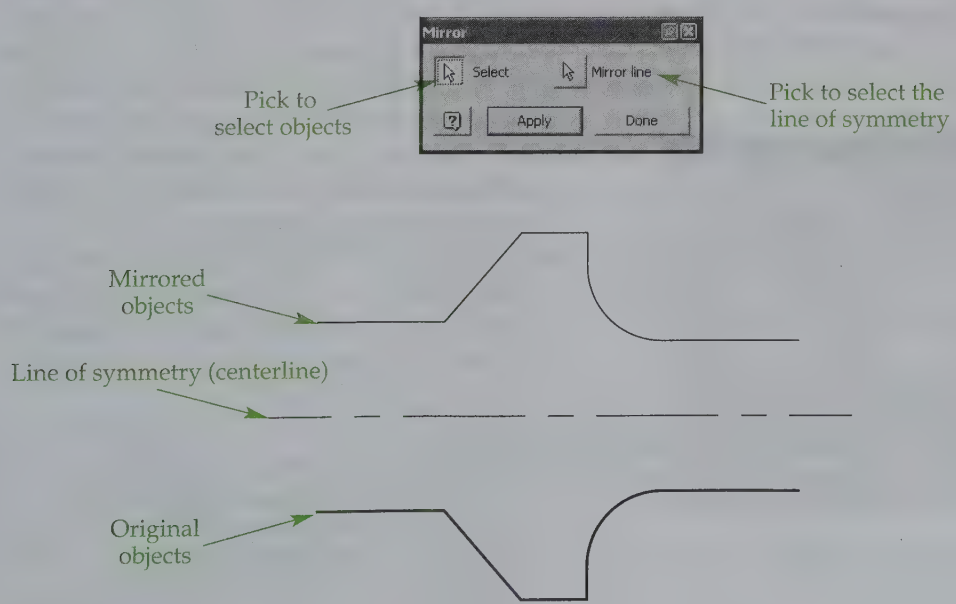


You may want to change an independent line of symmetry to a centerline to avoid confusing it with profile geometry. Centerlines are sketched exactly the same as lines. In order to change a normal or construction line to a centerline, you must choose the **Centerline** button from the **Inventor Standard** toolbar either before you access the **Line** tool, or after a line is drawn, by picking existing geometry and then selecting the **Centerline** button.



EXERCISE 4-14 Complete the Exercise on the Student CD.

Figure 4-25.
Use the **Mirror** tool and a line of symmetry to make a mirrored copy of objects.

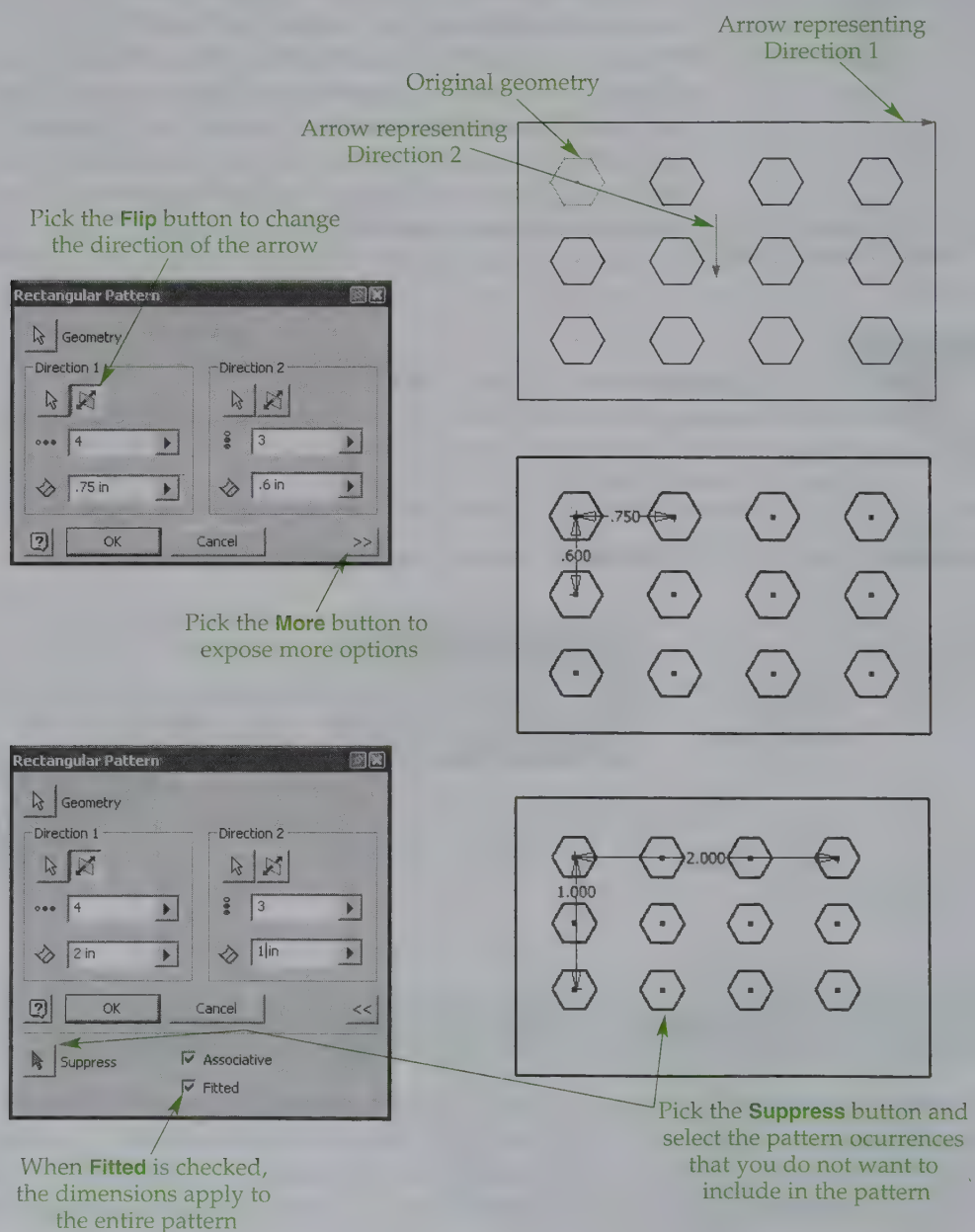


Creating Rectangular Sketch Patterns

Rectangular sketch patterns are created using the **Rectangular Pattern** tool. Access the **Rectangular Pattern** tool by picking the **Rectangular Pattern** button on the **2D Sketch Panel** bar or **2D Sketch Panel** toolbar.

The **Rectangular Pattern** dialog box is displayed as shown in Figure 4-26. To create a rectangular pattern of a sketch shape or group of shapes, first pick the **Geometry** button if it is not already active. Then, select the sketch geometry you want to array, pick the **Direction 1** button, located in the **Direction 1** area, and select the direction of the row or column of patterned geometry. You can select any available line, or sketch a construction line. When you choose a direction, an arrow shows you the specified pattern path, and a preview displays the pattern. If the direction is not correct, pick the **Flip** button to reverse the direction.

Figure 4-26.
The size of a rectangular pattern is controlled by the distance between instances or the distance of the overall pattern when using the **Fitted** option.



Once you define the direction of the first pattern, specify the total number of items you want to create by entering or selecting a count value using the **Count** edit box. Then define the distance between the copies by entering or selecting a spacing value using the **Spacing** edit box. Spacing is the distance between a point on one object to the corresponding point on the next object, not the clear space between items. For example, if you want to pattern a rectangle that is 1" (25mm) wide, and you want a 1/2" (12.5mm) space between pattern copies, you must specify a 1 1/2" (37.5mm) spacing. Another example is when creating a rectangular pattern of circles, and you want the distance between the circle centers to be 1" (25mm), specify a spacing of 1" (25mm).

Once you fully define **Direction 1**, repeat the steps previously described to define **Direction 2**. Pick the **Direction 2** button, located in the **Direction 2** area, and select the desired direction. If the first direction defines the rows, **Direction 2** defines the columns. If the first direction defines columns, **Direction 2** defines rows. You can select any available line, or sketch a construction line. You can choose the same line you chose for **Direction 1** or a parallel line. However, for most applications, you should select a line that is perpendicular, or at least nonparallel to the first direction. When you choose the second direction, an arrow shows you the specified pattern path, and a preview displays the pattern operation. If the direction is not correct, pick the **Flip** button to reverse the direction.

Once you define the direction of the second pattern, specify the total number of items you want to create by entering or selecting a count value using the **Count** edit box. Then define the distance between the copies by entering or selecting a spacing value using the **Spacing** edit box.

After you select the sketch geometry and specify direction, count, and spacing information, you may want to change the rectangular pattern creation method by picking the **More** button and accessing the additional options. Finally, pick the **OK** button to generate the rectangular pattern.

Creating Circular Sketch Patterns

Circular sketch patterns are created using the **Circular Pattern** tool. Access the **Circular Pattern** tool by picking the **Circular Pattern** button on the **2D Sketch Panel** bar or **2D Sketch Panel** toolbar.

The **Circular Pattern** dialog box is displayed when you access the **Circular Pattern** tool. See Figure 4-27. To create a circular pattern of a sketch or series of sketch shapes, pick the **Geometry** button if it is not already active. Then select the geometry you want to pattern. When all the sketched shapes are selected, pick the **Rotation Axis** button, and select the axis of rotation. The ***axis of rotation*** is the pivot point around which the selected geometry is copied. You can select any available point or create a sketch point for the axis of rotation. When you choose a rotation axis, an arrow shows you the specified pattern path, and a preview displays the pattern operation. If the direction is not correct, pick the **Flip** button to reverse the direction.

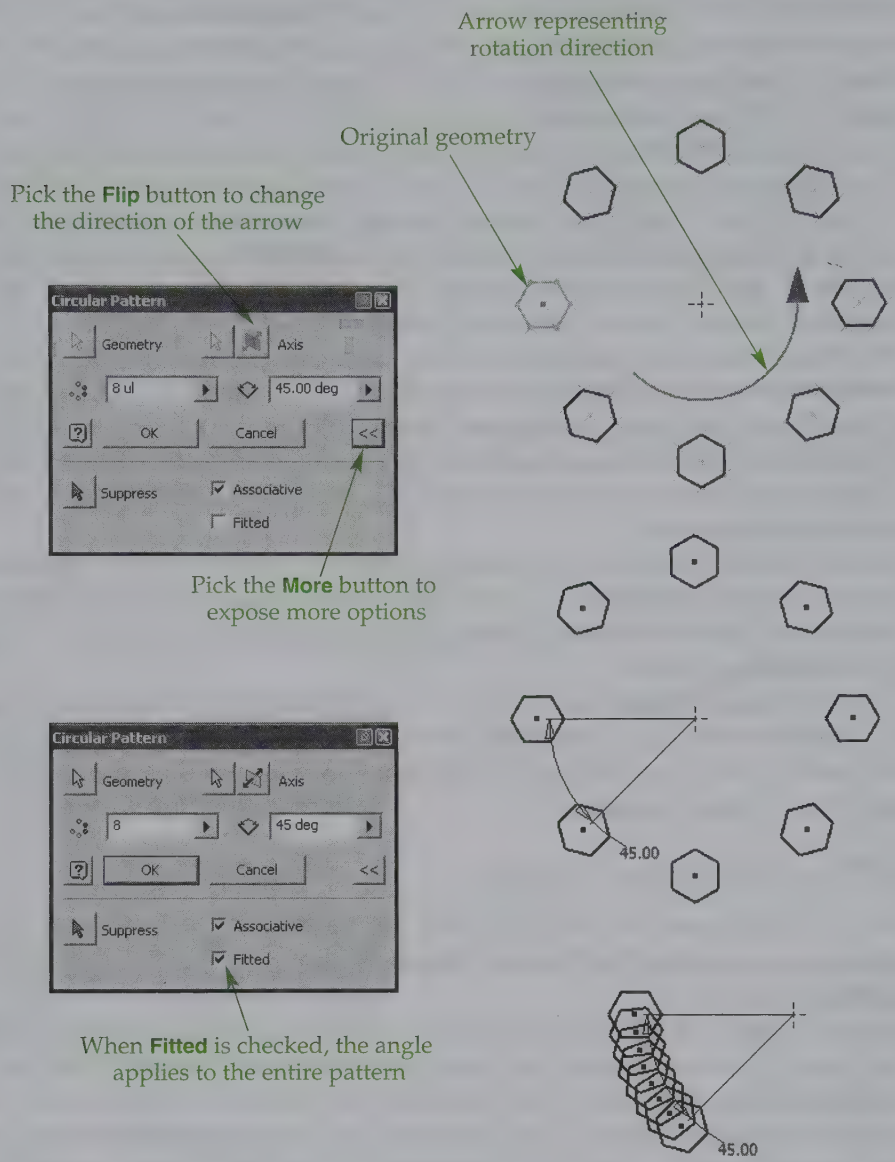
Once you select the geometry and define the axis of rotation, specify the total number of items you want to create by entering or selecting value using the **Count** edit box. Then you may want to change the circular pattern creation method by picking the **More** button and accessing the additional options. Finally, pick the **OK** button to generate the circular pattern.

NOTE



Dimensions corresponding to the pattern specifications are automatically placed when you pattern sketch geometry.

Figure 4-27.
A circular pattern is controlled by one count, one dimension (angle), and the three options that appear when the **More** button is selected.



PROFESSIONAL TIP



A ***sketch pattern*** is created anytime you place multiple arranged copies, or patterns, of sketches. Although typically features, not sketches, are patterned, you may find it helpful to pattern sketched shapes for some applications.



EXERCISE 4-15 Complete the Exercise on the Student CD.

Offsetting Sketch Geometry

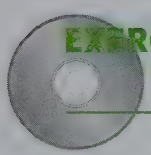
Use the **Offset** tool to *offset* most existing sketch geometry, including lines, splines, circles, arcs, ellipses, rectangles, fillets, and chamfers. The **Offset** tool is very effective for creating parallel objects a specified distance apart. See Figure 4-28. All geometric constraints associated with the parent object are duplicated in the offset object, if desired.

To offset sketch geometry, access the **Offset** tool by picking the **Offset** button on the **2D Sketch Panel** bar or **2D Sketch Panel** toolbar. Then select the objects you want to offset and drag the offset objects to the desired location. Notice the offset distance is displayed in the right side of the **Status** bar. When you are satisfied with the location of the offset, pick the location. Repeat the steps to offset additional sketch geometry, or press the [Esc] key, right-click and pick the **Done** option, or choose another tool.

NOTE

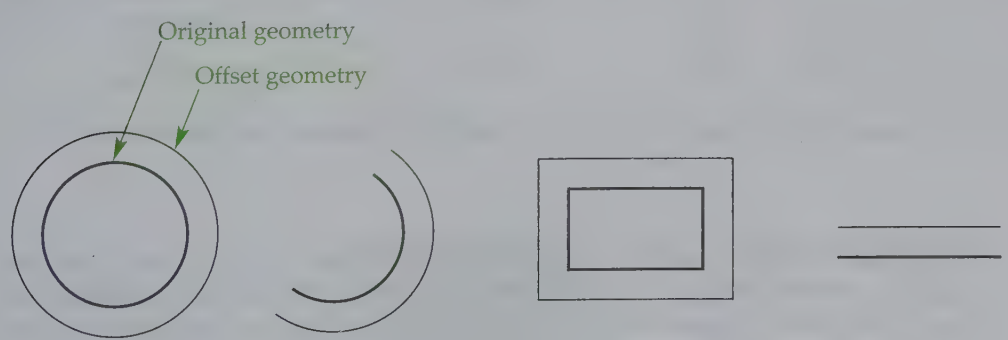


By default, when you offset objects, the entire loop is selected, as shown by the rectangle in Figure 4-28. The offset object is also constrained the same as the parent object. To select individual pieces of an object instead of the entire loop, such as a single rectangle line, while using the **Offset** tool, right-click and deselect the **Loop Select** option from the pop-up menu. Then, select the object or objects you want to offset, followed by right-clicking and picking the **Continue** pop-up menu option or pressing the [Enter] key. Finally, pick the offset location. If you do not want to apply the parent object constraints to the offset object, right-click and deselect the **Constrain Offset** option from the shortcut menu while using the **Offset** tool.



EXERCISE 4-16 Complete the Exercise on the Student CD.

Figure 4-28.
The **Offset** tool is used to create parallel objects.



Constraining Sketches Using Geometric Constraints

Geometric constraints, often just referred to as ***constraints***, are used to define sketch geometry in reference to other sketch geometry. Constraints are used for common geometric relationships, such as two perpendicular lines, equal-sized objects, or a line tangent to a circle. Usually, at least some geometric constraints are used to fully define a sketch. Autodesk Inventor provides you with the following options for constraining sketch objects:

- **Automatic constraints.** These constraints are applied to geometry as you sketch. You can choose which types of constraints are automatically available by selecting either the **Parallel and Perpendicular** radio button or the **Horizontal and Vertical** radio button in the **Constraint Placement Priority** area of the **Options** dialog box, **Sketch** tab. Selecting one of these radio buttons allows you to select a parallel, perpendicular, horizontal, or vertical point before any other point, such as a grid or snap coordinate.
  Automatic constraints are the same as the automatic referencing tools described in the *Sketching Lines* and *Splines* sections of this chapter. The endpoint, midpoint, center point, intersection, X,Y coordinates, tangent, and coincident options are all types of automatic constraints that create relationships between two or more objects. In addition, horizontal, vertical, parallel, concentric and perpendicular automatic constraints are also available. See **Figure 4-29.** To use one of these options, drag new sketch geometry until you see the appropriate automatic geometric constraint icon, and pick the location. To confirm that a constraint is present, attempt to drag the object. You should observe less freedom of movement.

NOTE

If you do not want automatic constraints to be placed on sketch geometry, hold down the [Ctrl] key while sketching. Some sketching tools automatically apply certain constraints to your sketch. For example, when you use the **Rectangle** tool to create a rectangle, parallel, perpendicular, coincident, and horizontal, constraints are added to the sketch to make a true rectangle.

Figure 4-29.
The icons indicate the presence and type of automatic constraints.



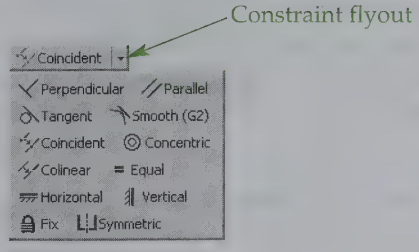
- **Constraint buttons.** Automatic geometric constraint tools are very effective for quickly constraining geometry while you sketch. However, you often need to apply additional constraints during sketch development using the Autodesk Inventor constraint buttons available by picking the desired constraint button on the **2D Sketch Panel** bar; picking the desired constraint button on **2D Sketch Panel** toolbar; right-clicking while in the sketch environment, and select the desired constraint from the **Create Constraint** cascading submenu; or picking the desired constraint button from the **2D Sketch Constraints** toolbar.

The active constraint button is displayed in the **2D Sketch Panel** bar and **2D Sketch Panel** toolbar. To access additional options, pick the flyout button and select the desired tool. See Figure 4-30. The following options are available inside the constraints flyout button:

- **Perpendicular.** This constraint allows you to create lines or ellipse axes that are *perpendicular*, or at a 90° angle, to each other. Unless the constraint is removed, you are not able to increase or decrease the angle between objects.
- **Parallel.** This constraint allows you to create objects that are parallel to each other. *Parallel* objects, such as lines and ellipse axes never intersect, no matter how long they become.
- **Tangent.** This constraint is used to define the tangent edge of one curve such as a line, spline, circle, arc, or fillet in reference to the tangent edge of an existing curve.
- **Smooth (G2).** This constraint is used to create a curvature-continuous situation, or G2 curve, between a spline and a line, second spline, or arc connected to the spline endpoint.
- **Coincident.** Use this tool to constrain two points, or a curve and a point, to the same location. The coincident constraint is often used to constrain initial sketch geometry to the projected center point.
- **Concentric.** Use this tool to constrain the center point of an ellipse, circle, or arc to the center point of another ellipse, circle, or arc.
- **Collinear.** This constraint is used to align two lines or ellipse axes along the same line.
- **Horizontal.** Use this tool to horizontally align lines, points, or ellipse axes. A horizontal constraint positions geometry along the X-axis on the XY plane, along the Y-axis on the YZ plane, and along the X-axis on the XZ plane.
- **Vertical.** Use this tool to vertically align lines, points, or ellipse axes. A vertical constraint positions geometry along the Y-axis on the XY plane, along the Z-axis on the YZ plane, and along the Z-axis on the XZ plane.
- **Equal.** This constraint allows you to size an object in reference to another object.
- **Fix.** This constraint allows you to secure a point or object to its current location in space.
- **Symmetric.** This constraint allows you to establish symmetry by selecting one object, followed by another object, and finally, a line of symmetry.

For constraint tools that require you to pick two objects, the first object you select always remains the same. The second object selected changes in relation to the initially selected reference object. For example, if you want to make two lines the same length using the **Equal** constraint tool, and select a 2" (50mm) line first, the second line you choose also becomes 2".

Figure 4-30.
Pick the flyout next to the **Constraint** tool to see the icons representing each constraint.



To constrain more geometry using the same tool, continue selecting objects. If you want to use a different constraint tool, access that tool and begin selecting objects. When you finish applying geometric constraints, press the [Esc] key, right-click and pick the **Done** option, or choose another tool.

NOTE

You can constrain sketch objects as previously described, or constrain a sketch or sketch object to a projected part feature edge, point, or axis.

EXERCISE 4-17

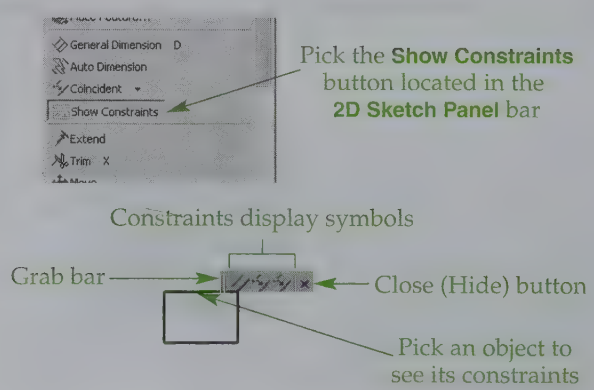
Complete the Exercise on the Student CD.

Using the Show Constraints Tool

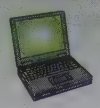
Often you may want to observe and delete constraints associated with a particular sketch object, so you can remove relationships or apply different constraints. The **Show Constraints** tool allows you to display all the geometric constraints placed on an object or group of objects. To show the constraints associated with only certain objects, pick the **Show Constraints** button located in the **2D Sketch Panel** bar or **2D Sketch Panel** toolbar, and select the desired object. All the established constraints are displayed, represented by their symbols. See Figure 4-31. You can select as many objects as desired using this technique. Notice that when you move your cursor over, or select a constraint symbol, the constraint and constrained objects become highlighted. To show all constraints applied to the entire sketch, without selecting individual objects, right-click on the browser bar or anywhere in the graphics window, and choose the **Show All Constraints** pop-up menu option or press the [F8] key.

When the constraint display symbols are visible, you have the option of deleting certain constraints in order to apply a different relationship. To delete constraints, right-click the constraint you want to remove and select **Delete** from the shortcut menu. If the constraint display symbols are blocking your view and need to be moved, pick the grab bar to the left of the symbols and drag the box to a new location. If you want to close a constraint display symbols box, pick the **Hide** button located to the right of the symbols. To close all the sketch constraints without selecting individual hide buttons, right-click on the browser bar or anywhere in the sketch environment graphics window, and choose the **Hide All Constraints** pop-up menu option or press the [F9] key.

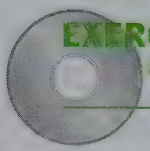
Figure 4-31.
You can display an object's constraints by picking the **Show Constraints** button and then selecting the object.



PROFESSIONAL TIP



Generally, you should apply geometric constraints before dimensions unless most constraints are automatically added. Placing too many constraints can cause problems as you progress through the design process. Apply only the constraints necessary to generate the particular sketch.



Complete the Exercise on the Student CD.

Constraining Sketches Using Dimensions

In addition to placing geometric constraints, another way to constrain a sketch is by applying dimensions. ***Dimensions*** are used to numerically define the size and location of sketch geometry, such as the length of a line, diameter of a circle, and radius of an arc. Usually, at least some dimensions must be used to fully constrain, or define, an object. Autodesk Inventor provides you with the **General Dimension** and **Auto Dimension** tools to help define sketch geometry.

PROFESSIONAL TIP



You know when your sketch is fully constrained because the object color changes and you are no longer able to drag any of the sketched items.

Using the General Dimension Tool

The **General Dimension** tool allows you to place aligned, linear, diameter, radius, or angular dimensions on a sketch by selecting individual objects. To apply a general dimension, access the **General Dimension** tool by picking the **General Dimension** button on the **2D Sketch Panel** bar; picking the **General Dimension** button on the **2D Sketch Panel** toolbar; right-clicking inside the graphics window or browser bar, and selecting the **Create Dimension** option; or pressing the [D] key. Next, pick the geometry you want to define with a dimension, and drag the dimension text to the desired location.

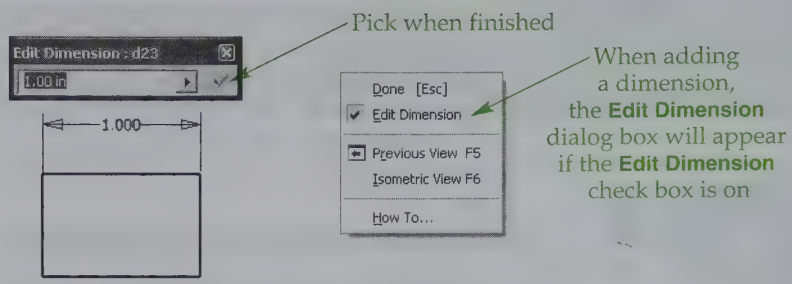
If you do not select the **Edit dimensions when created** check box in the **Sketch** tab of the **Options** dialog box, the object is dimensioned to its current measurement. You must then pick the displayed dimension to open the **Edit Dimension** dialog box, shown in Figure 4-32, and enter a value. If the **Edit dimensions when created** check box is selected, the **Edit Dimension** dialog box automatically appears when you create a dimension. You can also activate the **Edit dimensions when created** tool by right-clicking inside the graphics window while using the **General Dimension** tool, and selecting **Edit Dimension**.

NOTE



To edit sketch dimensions any time, even while not using the tool, double-click a dimension to access the **Edit Dimension** dialog box.

Figure 4-32.
The **Edit Dimension** dialog box, which allows you to specify a dimension, appears automatically when the **Edit Dimension** option of the **General Dimension** pop-up menu is selected.



There are a few different ways to specify dimension values in the **Edit Dimension** dialog box. You can type or select a specific numerical value using the edit box, or you can enter an equation. Usually equations are used when you do not know the exact numerical value of the dimension, or want your sketch to adapt according to dimension changes. Use the following keys to create an expression:

- The [+] key adds. For example, 2+2 or 3in+5mm.
- The [-] key subtracts. For example, 5-2 or 7in-5mm
- The [*] key multiplies. For example, 5*2 or 7in*5mm
- The [/] key divides. For example, 6/2 or 7in/5mm
- The [^] key allows for power operations. For example 3^5.
- The parentheses keys [(] and [)] separate expressions. For example, (5*3)+(47.25/26).
- In addition to these standard operations, you can also use functions such as sin, cos, log, and tan.

NOTE



Dimensions defined using equations are differentiated from other dimensions with fx: before the dimension value. For example, a dimension equation of 2+2 is displayed as fx:4 in the sketch.

PROFESSIONAL TIP



You can enter equations for most Autodesk Inventor edit boxes. You may want to refer to the **Equations** reference section of the Autodesk Inventor help files for further information regarding expressions, equation prefixes, units, and procedures for entering more complicated expressions.

Dimension units reflect the current work environment and unit settings, such as inch or metric. Therefore, the value entered is represented in these units. For example, typing 1 means 1" when working with inch units. However, you can also enter units after a numerical value to convert the value to the specified units. For example, you can enter 20mm even while in an inch-unit system.

Another way to specify a dimension value is to reference existing sketch dimensions. To apply this technique, access the **Edit Dimension** dialog box, followed by selecting the existing dimension you want to reference. Notice that the title of the selected

dimension is displayed in the edit box. See Figure 4-33. If you pick the check mark, the object becomes the same size as the reference object. It is easier and more effective to use an **Equal** geometric constraint for this process. As a result, usually when you reference an existing dimension to define an object, you include an equation, as shown in Figure 4-34.

The **General Dimension** tool has a number of options that allow you to dimension and define the size and location of any type of sketch geometry. The following describes dimensioning options available using the **General Dimension** tool:

- **Linear dimensions.** These dimensions are used to define the vertical and horizontal size and location of objects. To place a linear dimension, access the **General Dimension** tool and pick the object, or two object points, you want to dimension. If the object is vertical, drag the dimension to the left or right and pick a location. If the object is horizontal, drag the dimension up or down, and pick a location. See Figure 4-35. If the object can be dimensioned by a vertical or horizontal dimension, right-click to select **Vertical** or **Horizontal** from the

Figure 4-33.
When adding a new dimension, you may pick an existing dimension to create a reference.

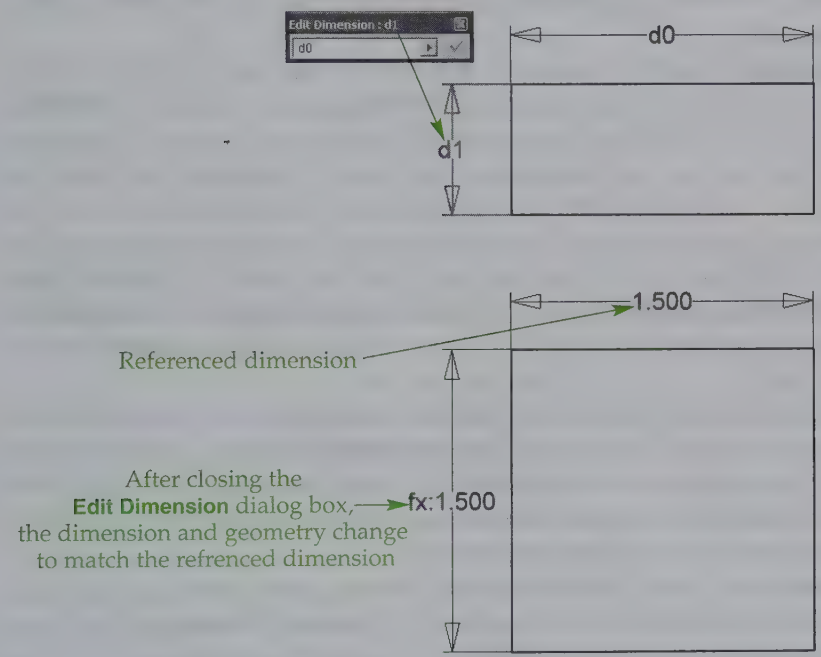


Figure 4-34.
Using existing dimensions and equations to dimension an object.

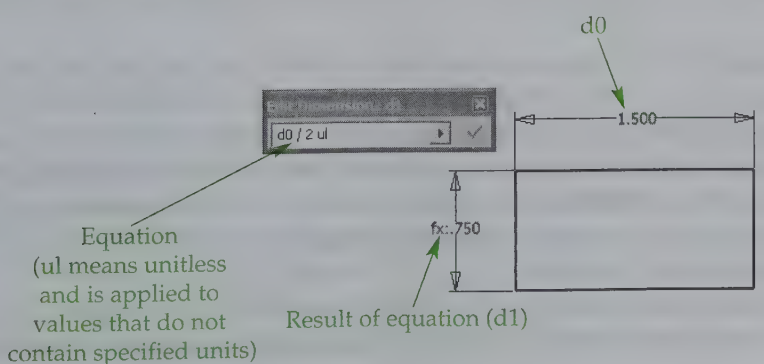
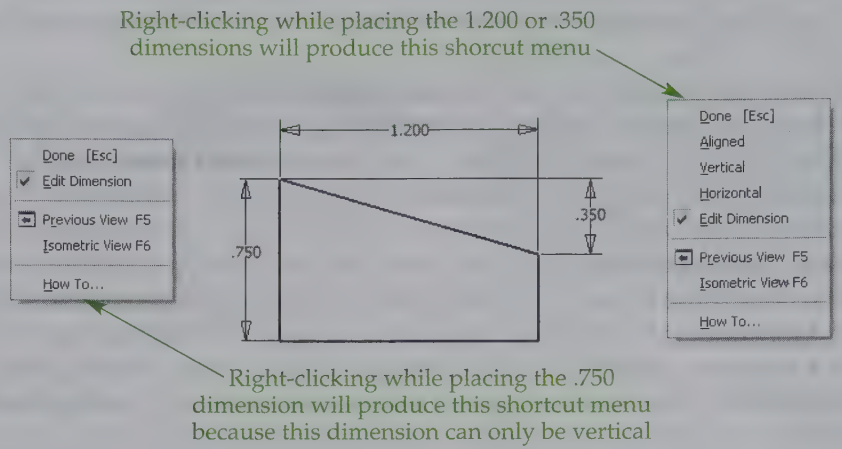


Figure 4-35.
Linear sketch dimensions, such as these, define the horizontal and vertical sizes and locations.



shortcut menu before placing the dimension. This process helps to isolate a vertical or horizontal dimension placement. Repeat the steps to create more linear dimensions, or press the [Esc] key, right-click and pick the **Done** option, or select another tool.

- **Aligned dimensions.** These dimensions are used to define an object that is not vertical or horizontal. To place an aligned dimension, access the **General Dimension** tool and pick the object, or two object points, you want to dimension. The default setting for placing dimensions is linear. As a result, to create an aligned dimension, you must select the object a second time or pick while the **Aligned Dimension** icon is showing. See Figure 4-36. Then, drag the dimension to the desired location and pick. You can also pick the object, right-click, and select **Aligned** from the shortcut menu. Create another aligned dimension by repeating the steps, or press the [Esc] key, right-click and pick the **Done** option, or pick another tool.
- **Angular dimensions.** These dimensions define the angle between two lines. To place an angular dimension, access the **General Dimension** tool, and select two lines. Then, drag the dimension to the desired location, and pick. See Figure 4-37. Continue drawing additional angular dimensions, or press the [Esc] key, right-click and pick the **Done** option, or select another tool.
- **Diameter dimensions.** These dimensions are used to define the diameter of a circle. To place a diameter dimension, access the **General Dimension** tool and select a circle. Then drag the dimension to the desired location and pick. See Figure 4-38. If for some reason you want to dimension a circle as a radius instead of a diameter, select the circle, right-click, and choose **Radius** from the shortcut menu. To create another diameter dimension, repeat the steps. If you are finished, press the [Esc] key, right-click and pick the **Done** option, or select another tool.
- **Radius dimensions.** These dimensions are used to define the radius of an arc. To place a radius dimension, access the **General Dimension** tool and select an arc. Then drag the dimension to the desired location and pick. See Figure 4-39. If for some reason you want to dimension an arc as a diameter instead of a radius, select the arc, right-click, and choose **Diameter** from the shortcut menu. Create another radius dimension by repeating the steps, or press the [Esc] key, right-click and pick the **Done** option, or choose another tool.
- **Driven dimensions.** When you place a dimension on a sketch object that is already dimensioned, the sketch should become over-constrained. However,

Figure 4-36.
There are three methods to apply an aligned sketch dimension.

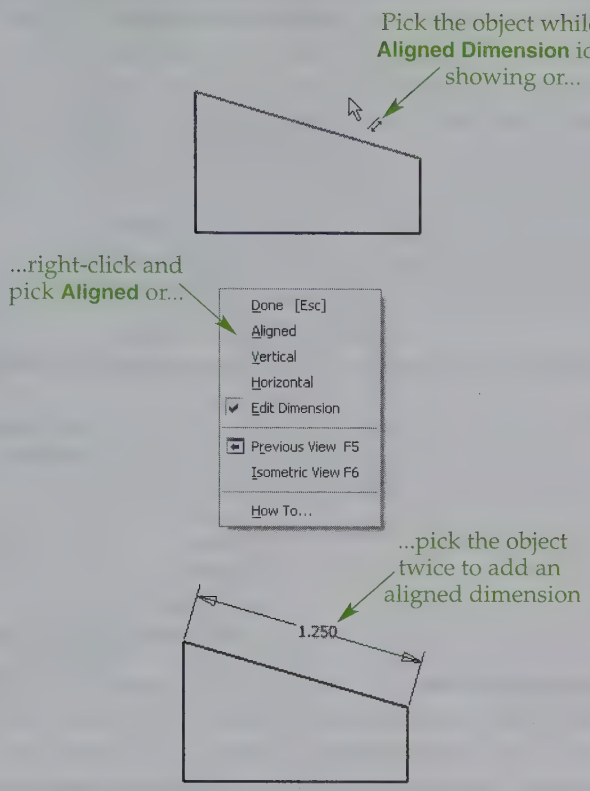
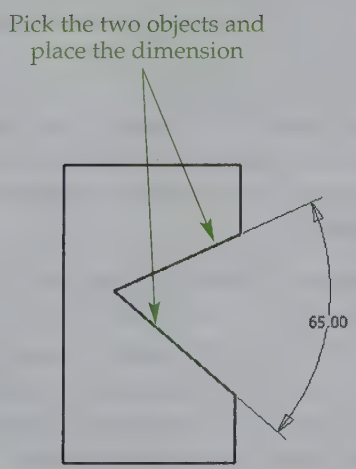


Figure 4-37.
Applying an angular sketch dimension requires two nonparallel lines.



it is not possible to over-constrain a sketch, because Autodesk Inventor does not allow over-constraining to occur. As a result, you either do not accept the dimension and cancel the tool, or accept the additional dimension and allow the dimension to become ***driven***. Driven dimensions are identified by being enclosed in parentheses. See **Figure 4-40.** Driven dimensions are used for reference and are not parametric. You cannot edit a driven dimension to change the size of an object, but a driven dimension changes if the corresponding

Figure 4-38.
Circles should always be dimensioned by the diameter, though it is possible to dimension circles with a radius.

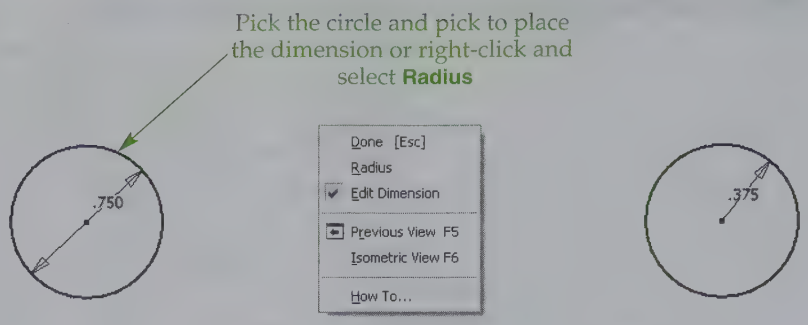


Figure 4-39.
Arcs should always be dimensioned by the radius, though it is possible to dimension arcs with a diameter.

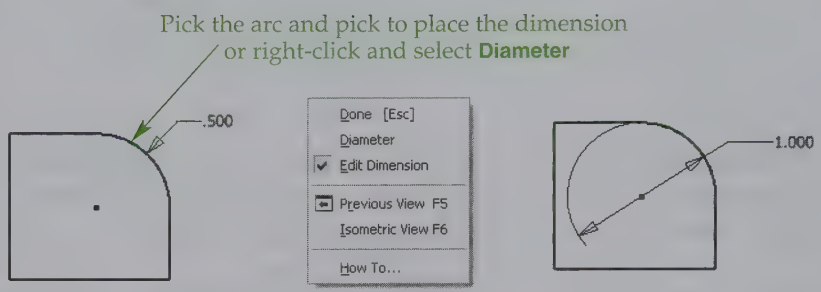
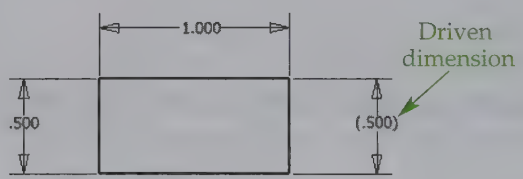


Figure 4-40.
The driven dimension, identified by the parentheses, is for reference only.



parametric dimension is modified. If you do not want Autodesk Inventor to warn you of an over-constrained condition, pick the **Apply Driven Dimension** radio button, which is found in the **Overconstrained Dimensions** area of the **Options** dialog box, **Sketch** tab.

PROFESSIONAL TIP



You can create a driven dimension without attempting to over-constrain a sketch. Usually these types of driven dimensions are used only for reference when creating an adaptive part. A non-driven, parametric dimension is created in the **Normal** style. If you want to change a non-driven dimension to driven, select the **Driven Dimension** button of the **Inventor Standard** toolbar.

NOTE

Sketch dimensions are used to define and constrain sketch geometry. Sketch dimensions are created using Autodesk Inventor and may not comply with ASME standards. Do not be overly concerned about the placement or display characteristics of these dimensions, but if possible apply sketch dimensions just as you would add dimensions to a drawing using correct drafting practices. Also, you may want to move and manipulate sketch dimensions so the sketch environment is as uncluttered as possible. To move dimensions, select the dimension you want to move and drag it to a new location.

EXERCISE 4-19 Complete the Exercise on the Student CD.

Using the Auto Dimension Tool

The **Auto Dimension** tool allows you to automatically dimension and constrain several objects at once. To access the **Auto Dimension** tool and open the **Auto Dimension** dialog box shown in Figure 4-41, pick the **Auto Dimension** button on the **2D Sketch Panel** bar or **2D Sketch Panel** toolbar.

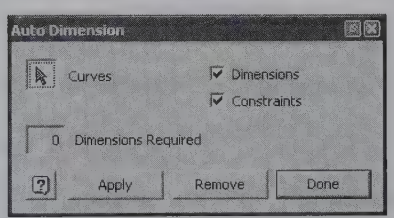


Figure 4-41.
The **Auto Dimension** dialog box is useful for adding dimensions and constraints. It is also a good way to determine how many dimensions are required to fully constrain the sketch.

You can pick specific objects when the **Curves** button is on, or select all of them by picking the **Apply** button. When using the **Auto Dimension** tool, deselect the **Dimensions** or **Constraints** check box if you do not want them added. Inventor lets you know how many more dimensions are required to fully constrain the sketch. This number is in the **Dimensions Required** box.

The number of dimensions required to fully constrain the sketch is displayed in the **Dimensions Required** box. By default, if you immediately pick the **Apply** button, the total number of dimensions and constraints are applied to the sketch. However, if you want to select individual objects, pick the **Curves** button and pick only the geometry you want to constrain with the **Auto Dimension** tool. The **Dimensions** and **Constraints** check boxes are selected by default. Deselect the **Dimensions** check box if you do not want to dimension the selected geometry, and deselect the **Constraints** check box if you do not want to geometrically constrain the selected geometry. Pick the **Remove** button to remove dimensions and constraints already placed on the sketch. Pick the **Done** button when finished using the **Auto Dimension** tool.

NOTE

The dimensions you create using the **General Dimensions** tool are not replaced when you use the **Auto Dimension** tool.

PROFESSIONAL TIP

The **Edit dimensions when created** check box of the **Options** dialog box, **Sketch** tab has no function in the **Auto Dimension** tool. Therefore, dimensions placed using the **Auto Dimension** tool represent the current object measurements. To modify a dimension value, you must double-click on a dimension and enter a value in the **Edit Dimension** dialog box.

NOTE

You can dimension sketch objects as previously described, or dimension a sketch or sketch object in reference to an existing part feature edge.

EXERCISE 4-20

Complete the Exercise on the Student CD.

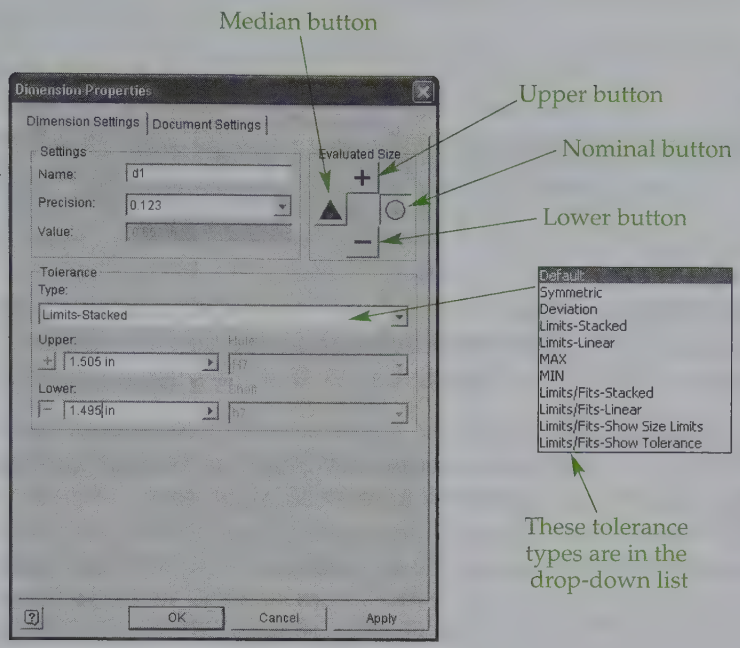
Working with Sketch Dimension Properties

All sketch dimensions added to a sketch contain specific individual dimension properties. Most of these properties are predefined by default, or specified in the **Document Settings** dialog box, and typically do not need to be individually altered. Still, you can examine and manipulate individual dimensions as needed using the **Dimension Properties** dialog box, shown in Figure 4-42.

Access the **Dimension Properties** dialog box by right-clicking on any sketch dimension and select the **Dimension Properties...** pop-up menu option. The **Dimension Properties** dialog box contains the following two tabs:

- **Dimension Settings.** This tab is used to override the existing dimension settings of the selected dimension. Inside this tab you find the following three areas:
  - **Settings.** As you will learn, each model item throughout the design process, including sketch dimensions, is given a name. You can modify the default name of the selected dimension to something more meaningful to the sketch, by entering a new name in the **Name:** edit box. For example, you may want to change the name of d1 to Block_Width. In addition, you can override the dimension value precision by selecting a different precision from the **Precision:** drop-down list, and view the dimension value in the **Value:** display box.
  - **Evaluated Size.** This area sets the tolerance level of the selected dimension, with the **Nominal** radio button selected as default. *Nominal* is the normal value of a commercial product and is intended to be the true drawn size without any specified limits. Selecting either the **+ (Upper)**, **– (Lower)**, or **Median** button changes the dimension to be set at the designated upper,

Figure 4-42.
The **Dimension Properties** dialog box with the **Dimension Settings** tab displayed.



lower, or median tolerance and places a line under the dimensional value, which means "not to scale".

- **Tolerance.** This area allows you to set tolerance values for the selected dimension. The options in the **Tolerance** area are available only when a specific tolerance method is selected. The **Lower** and/or **Upper** edit boxes are available when you define limit tolerance methods such as **Symmetrical**, **Deviation**, **Limits – Stacked**, and **Limits – Linear**. Enter a value in the **Upper** edit box to set an upper value for a tolerance with a value range. For example, an object that is 8" long and has an upper range of .05", has an upper tolerance limit of 8.05" (8+.05 = 8.05). Similarly, you can enter a value in the **Lower** edit box to set a lower value for a tolerance with a value range. Using the previous example, an object with a lower range of .05 in. has a lower tolerance limit of 7.95" (8 – .05 = 7.95). The tolerance for this dimension reads 8 ±.05 for a **Symmetric** tolerance method option.

PROFESSIONAL TIP



When using the **Deviation** tolerance method option, both the **Upper** and **Lower** edit boxes are available. The normal **Deviation** application is used to set up a bilateral tolerance such as .625±.005 or .625+.005/–.002. To accomplish this, you need to enter a minus (–) symbol in front of the value in the **Lower** edit box. A positive symbol is automatically applied to a positive number without adding a plus (+) sign.

When using limits and fits tolerance methods for dimensioning holes and shafts, the **Hole** and **Shaft** drop-down lists are available. The hole values—A1 or B2, for example—are available in the **Hole** drop-down list corresponding to specific hole tolerances established in ANSI size and fit tables. For example, a hole dimensioned 20 H9 takes the place of 20 + 0.05/0. In the same respect as the **Hole** drop-down list, the shaft values—a1

or b2, for example, are available in the **Shaft** drop-down list corresponding to specific hole tolerances established in ANSI size and fit tables. For example, a shaft dimensioned 10d9 takes the place of 9.950/–0.05.

NOTE

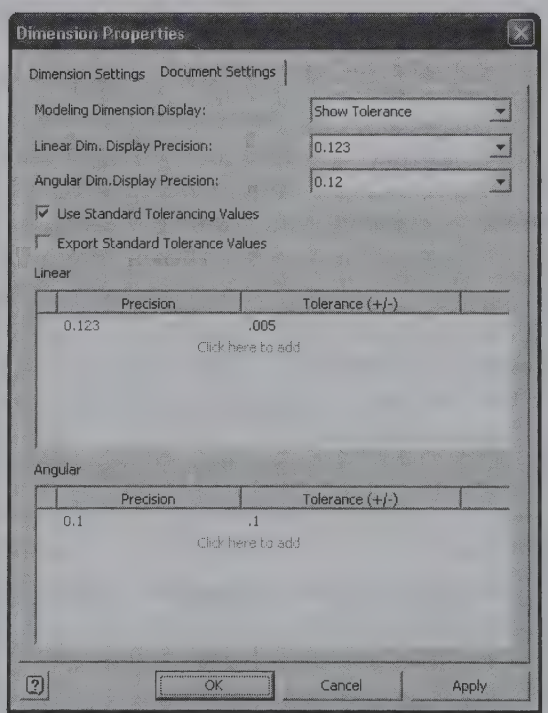
When you select a value from the **Hole** drop-down list, the **Shaft** value is N/A. Similarly, when you select a value from the **Shaft** drop-down list, the **Hole** value is N/A because the ANSI size and fit tolerances apply to either the hole or shaft.

- **Document Settings.** This tab, shown in Figure 4-43, contains many of the same options found in the **Document Settings** dialog box. Use this tab to change the document settings for the entire file, not just the selected dimension.

NOTE

You can also modify many individual dimension settings using the **Parameters** dialog box.

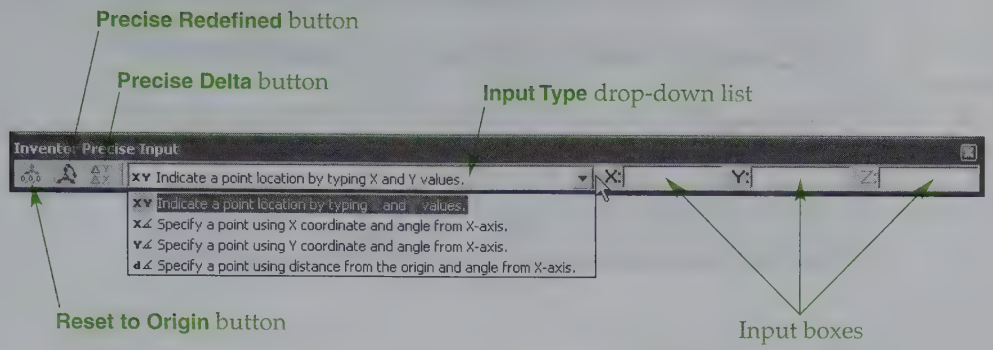
Figure 4-43.
The **Dimension Properties** dialog box with the **Document Settings** tab displayed



Sketching with the Precise Input Toolbar

The **Inventor Precise Input** toolbar, shown in Figure 4-44, has a number of options that allow you to identify the size of sketch geometry when it is initially created. For example, you can define the exact start or endpoint of a line, center point of a circle, the

Figure 4-44.
The **Inventor Precise Input** toolbar can be used to locate points while sketching objects.



points of a three-point arc, or the corners of a rectangle using points.

Using precise input techniques allows you to define the initial location of objects. However, using precise input tools does not constrain or dimension sketch geometry. In order to constrain the sketch you must still place dimensions and geometric constraints. As a result, most of the time, using precise inputs to create objects takes longer and is more complicated than drawing a general sketch and placing dimensions and geometric constraints. Still, the **Inventor Precise Input** toolbar can be effective for some applications.

To access the **Inventor Precise Input** toolbar, pick **View>Toolbar>Inventor Precise Input** or right-click on any existing toolbar and select the **Inventor Precise Input** menu option. You must activate a sketching tool, such as **Line**, **Arc**, or **Circle**, to use the **Inventor Precise Input** toolbar.

NOTE



In addition to standard numerical values, you can enter equations in most Autodesk Inventor edit boxes, including the **Input** edit boxes.

PROFESSIONAL TIP

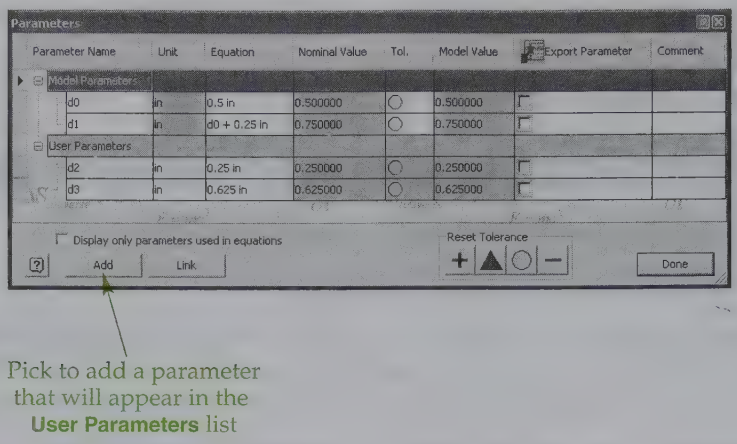


Precise input units reflect the current work environment and unit settings. However, you can enter units after a numerical value to convert the value to the specified units.

Sketch Parameters

Parameters are shape and size limits placed on sketches and features. Parameters in the sketch environment refer to the dimensions placed on sketch geometry. A model parameter is automatically created every time you dimension a sketched object. To access the **Parameters** dialog box/spreadsheet, select the **Parameters...** button on the **2D Sketch Panel** toolbar or **2D Sketch Panel** bar; or pick **Tools>Parameters...** Figure 4-45 shows the model parameters for a sketch with four dimensions. The following information is displayed under the **Model Parameters** node of the **Parameters** dialog box:

Figure 4-45.
The **Parameters** dialog box is a good place to review, edit, and export parameters.



- **Parameter Name.** This edit box displays the default parameter name. The first dimension applied to a sketch receives the name d0, the next dimension d1, and so on. To change a default parameter name to something meaningful to the sketch, such as Length, Width, or Diameter, pick inside the edit box. Then type the desired name and press the [Enter] key, or select the **Done** button if you are finished using the **Parameters** dialog box.
- **Unit.** This display box shows the measurement units used to dimension the object.
- **Equation.** This edit box displays the value or equation specified when dimensioning a sketch. To change an equation, pick inside the edit box and type the desired value or equation. Then press the [Enter] key, or select the **Done** button if you are finished using the **Parameters** dialog box.
- **Nominal Value.** This display box shows the nominal value of the dimensioned object. Again, the nominal value is intended to be the true drawn object size without any specified tolerance limits.
- **Tolerance.** This drop-down list allows you to select either a nominal, lower, upper, or median tolerance setting for the selected dimension.
- **Model Value.** This display box shows the current parameter value. If the dimension was created using an equation, the calculated value is displayed.
- **Export Parameter.** When you pick this check box, the selected dimension is added to custom properties for use in applications such as parts lists and a bill of materials.
- **Comment.** This edit box allows you to enter a parameter comment. To create a comment, pick inside the edit box and type the desired information. Then press the [Enter] key, or select the **Done** button if you are finished using the **Parameters** dialog box.
- **Reset Tolerance.** This area contains **Upper**, **Median**, **Nominal**, and **Lower** buttons that allow you to define or reset all the parameter tolerances in the **Parameters** dialog box to either upper, median, nominal, or lower.
- **Display only parameters used in equations.** Select this check box if you want to show only parameter values that are equations, and values associated with equations.
- **Add.** This button is used to add user-defined parameters.
- **Link.** Pick this button to link or embed a specified spreadsheet to use for parameters.

In addition to the automatically specified model parameters, you can also specify your own parameters in the **User Parameters** section of the **Parameters** dialog box by selecting the **Add** button previously mentioned. User parameters function just like model parameters in the **Parameters** dialog box.

PROFESSIONAL TIP

You may find it helpful to create parameters in the **User Parameters** section of the **Parameters** dialog box if you want access to specific parameters throughout the design process. For example, if you establish user parameters, when you select the cascading submenu next to a variety of edit boxes, the **List Parameters** option is available, which allows you to enter a selected user parameter in the edit box.

NOTE

Parameters and the **Parameters** dialog box can be used throughout the design process, from a simple sketch to the most complex assembly.

EXERCISE 4-21

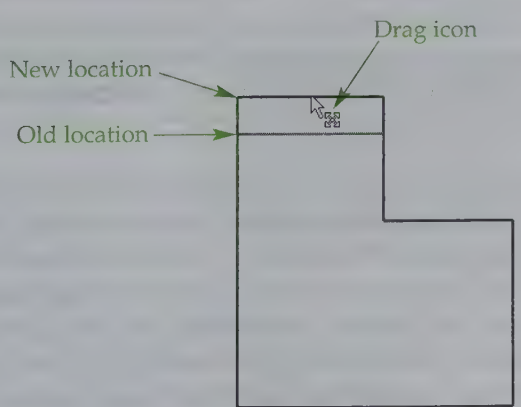
Complete the Exercise on the Student CD.

Using Sketch Editing Tools

Autodesk Inventor provides you with a number of sketch editing tools. Use these tools during and after sketch creation to modify geometry. Some of these sketch editing tools, such as modifying line, dimension, and point style using the **Construction**, **Centerline**, **Center Point**, and **Driven Dimension** tools of the **Inventor Standard** toolbar have already been described. The following information identifies additional sketch editing options:

- **Dragging geometry.** Sketch objects that are not constrained can be dragged to a new location or size. To drag an object, pick and hold down the left mouse button on the object. The drag icon, shown in Figure 4-46, becomes visible.

Figure 4-46.
The drag icon appears when you drag sketch geometry.



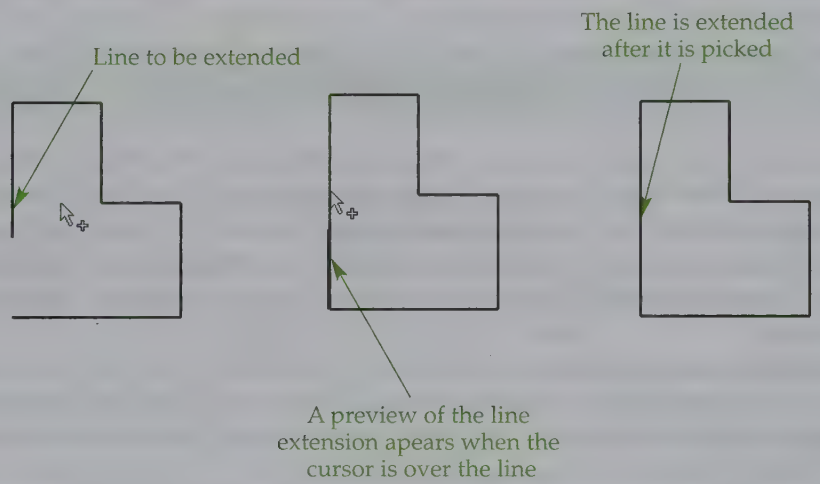
Then move the object to the desired location and release the left mouse button. Often, dragging or attempting to drag an object allows you to see where a constraint is still required, and assess design options.

NOTE

You can use all the automatic geometry referencing tools previously described in the **Line** tool when dragging objects.

- **Extend.** Use this button to increase the size of a curve, such as a line, spline, or arc, to the nearest intersection. In order to create a feature, a sketch must have a closed loop, which means it does not contain any gaps. You can often close a gap by using the **Extend** tool to extend a curve to the nearest intersection. To extend a curve, as shown in Figure 4-47, access the **Extend** tool by picking the **Extend** button on the **2D Sketch Panel** bar or **2D Sketch Panel** toolbar. Then, move your cursor over the object you want to extend, displaying a preview of the operation. If the preview looks acceptable, pick the curve to complete the extension. If desired, pick a new curve to extend, or press the [Esc] key, right-click and pick the **Done** option, or pick another tool.

Figure 4-47.
The **Extend** tool extends the selected curve to the nearest intersection or apparent intersection.

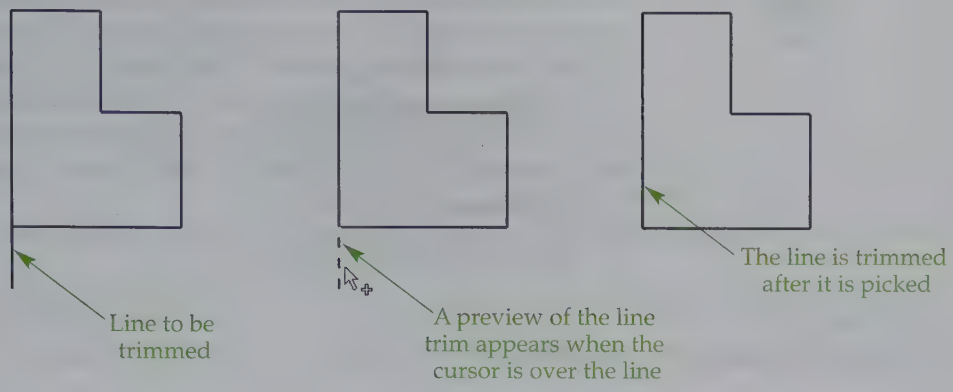


NOTE

If you just extended a curve and now need to trim a segment from your sketch, press the [Shift] key to access the **Trim** tool, or right-click and choose the **Trim** option.

- **Trim.** Use this button to remove unwanted sketch objects that extend past an intersection. See Figure 4-48. To trim an object, access the **Trim** tool by picking the **Trim** button on the **2D Sketch Panel** bar or **2D Sketch Panel** toolbar. Then move your cursor over the object you want to remove, displaying a preview of the operation. If the preview looks acceptable, pick the sketch segment to complete the trim. If desired, pick a new object to trim. If you do not want to extend another object, press the [Esc] key, right-click and pick the **Done** option, or pick another tool.

Figure 4-48.
The **Trim** tool trims the selected curve to the nearest intersection or apparent intersection.

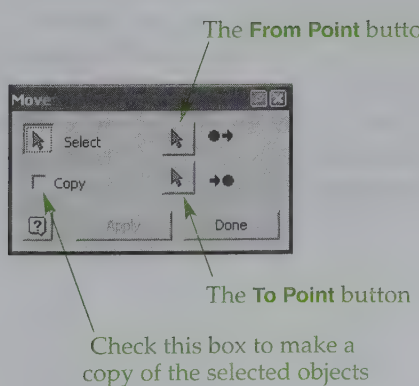


NOTE

If you just trimmed an object and now need to extend a curve in your sketch, press the [Shift] key to access the **Extend** tool, or right-click and choose the **Extend** option.

- **Move.** Use this tool to move an entire sketch or individual sketch objects from one location to another. To move an object, access the **Move** tool by picking the **Move** button on the **2D Sketch Panel** bar or **2D Sketch Panel** toolbar. Selecting the **Move** button opens the **Move** dialog box, shown in Figure 4-49. Now, select the objects you want to move, by picking them individually or using a crossing window. Once all the geometry you want to move is selected, right-click and pick the **Continue** option, or pick the **From Point** button of the **Move** dialog box, and select the base point from which you want to move the objects. Once the *from* point is selected, the **To Point** button is activated, allowing you to select the endpoint, to where you want to move the objects. If you want to copy the objects in addition to moving them to a new location, pick the **Copy** check box. Once the *to* point is selected and all other options are specified, press the [Enter] key or pick the **Apply** button to move the objects.

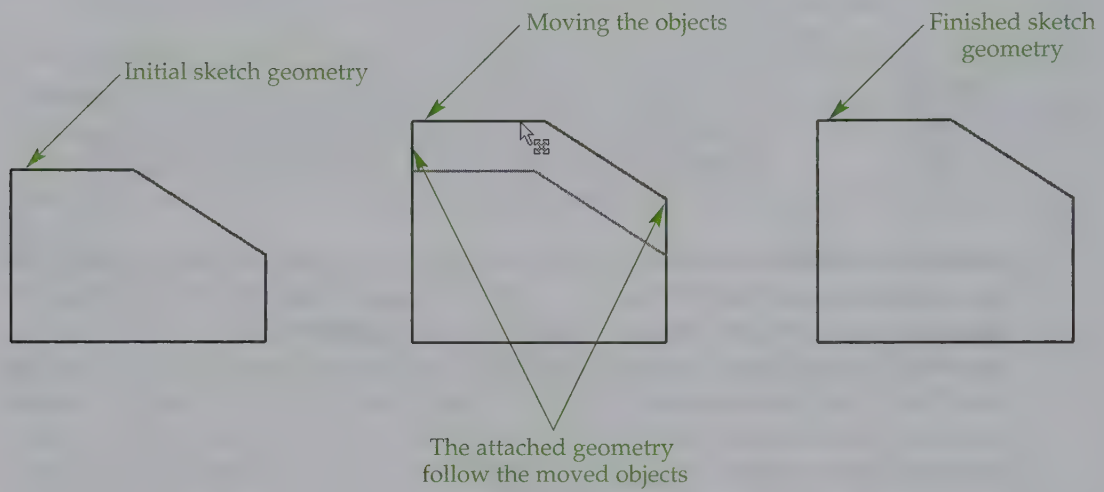
Figure 4-49.
The **Move** dialog box is also used to copy items.



NOTE

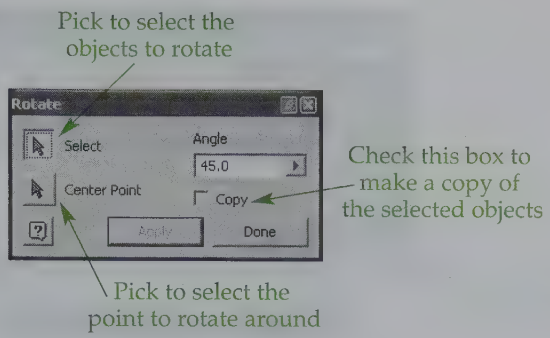
When you move individual sketch objects, those pieces of the sketch connected to the object being moved change to accept the new location of the object, but only if constraints are placed on the sketch geometry. See Figure 4-50. This process is similar to a stretch tool. You can also copy sketch geometry by selecting the object you want to duplicate, right-clicking, and choosing the **Copy** option. Then right-click again, and pick the **Paste** option.

Figure 4-50.
Moving sketched geometry stretches any other geometry to which it is constrained.



- **Rotate.** Use this tool to rotate an entire sketch or individual sketch objects from one location to another. To rotate an object, access the **Rotate** tool by picking the **Rotate** button on the **2D Sketch Panel** bar or **2D Sketch Panel** toolbar. This opens the **Rotate** dialog box, shown in Figure 4-51. Now select the objects you want to rotate by picking them individually or using a crossing window. Once all the geometry you want to move is selected, right-click and pick the **Continue** option, or pick the **Center Point** button of the **Rotate** dialog box, and select the center, or base, point you want to rotate the objects around. Once you identify the center point, enter the desired rotation angle in the **Angle** edit box, or select an angle from the drop-down list. If you want to copy the objects in addition to rotating them, pick the **Copy** check box. Once all your rotation options are specified, press the [Enter] key or pick the **Apply** button to rotate the designated objects.

Figure 4-51.
The **Rotate** dialog box is used to rotate original sketch items and rotate copies of sketch items.



NOTE

When you rotate individual sketch objects, the sketch objects connected to the object being rotated change to accept the new location of the object, but only if constraints are placed on the sketch geometry.

EXERCISE 4-22

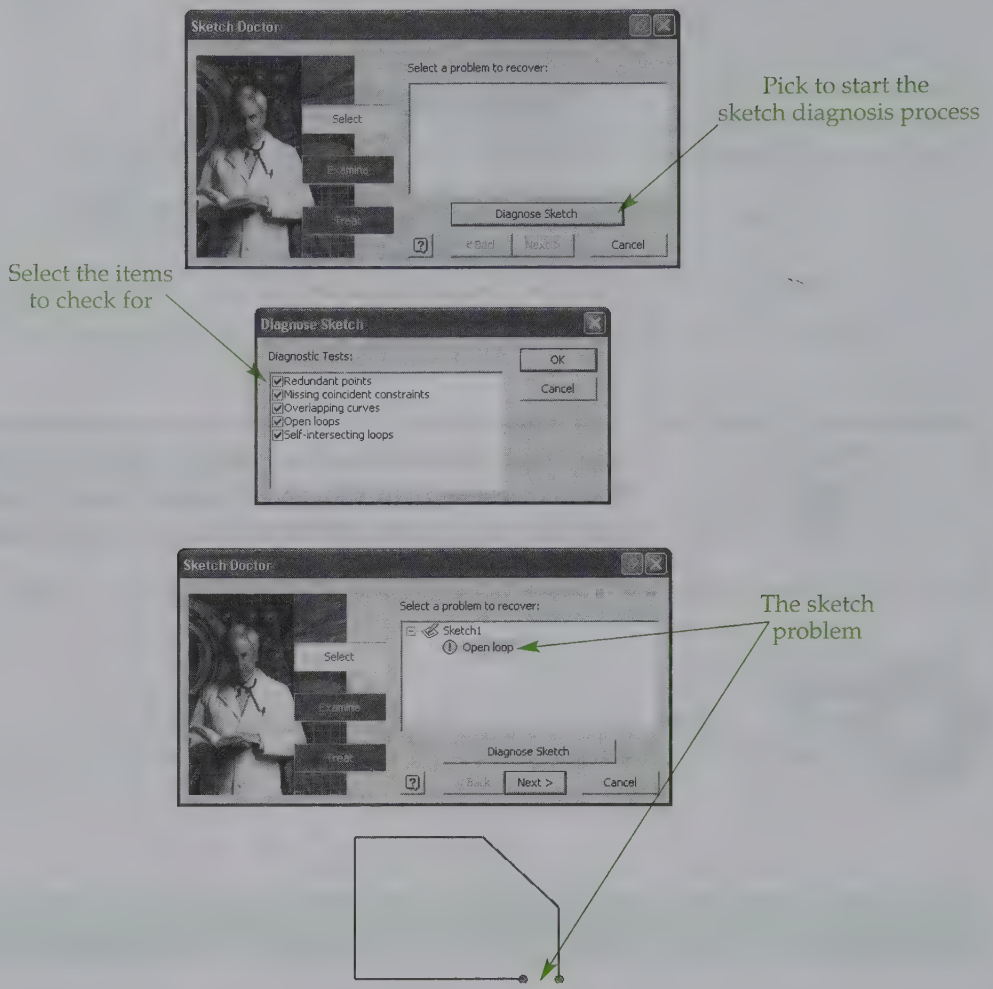
Complete the Exercise on the Student CD.

Using the Sketch Doctor

Once you create a sketch, the next step in developing a model is to produce a feature from the sketch. In order to create a feature, most of the time your sketch should result in a closed loop and be free of defects. Sketch problems or defects are typically the result of unneeded points, a missing coincident constraint, overlapping curves, open loops, or self-intersecting loops. The Sketch Doctor is a tool that helps you resolve these types of sketch problems. Access the Sketch Doctor by right-clicking anywhere in the graphics window or browser bar, and selecting the **Sketch Doctor** option while in the sketch environment; picking the **Recover** button on the **Inventor Standard** toolbar; or when creating a sketched feature, such as an extrusion, picking the **Examine Profile Problems** button on the sketch feature dialog box.

If you access the Sketch Doctor from a sketched feature dialog box, you automatically see the **Examine** section of the **Sketch Doctor** dialog box. Otherwise, the **Select** section is active. See Figure 4-52. Here you can choose the **Diagnose Sketch** button to open the **Diagnose Sketch** dialog box. The **Diagnose Sketch** dialog box allows you to choose which of the previously mentioned diagnostic tests to use. By default, all the diagnostic

Figure 4-52.
Use the Sketch Doctor to find problems with a sketch.



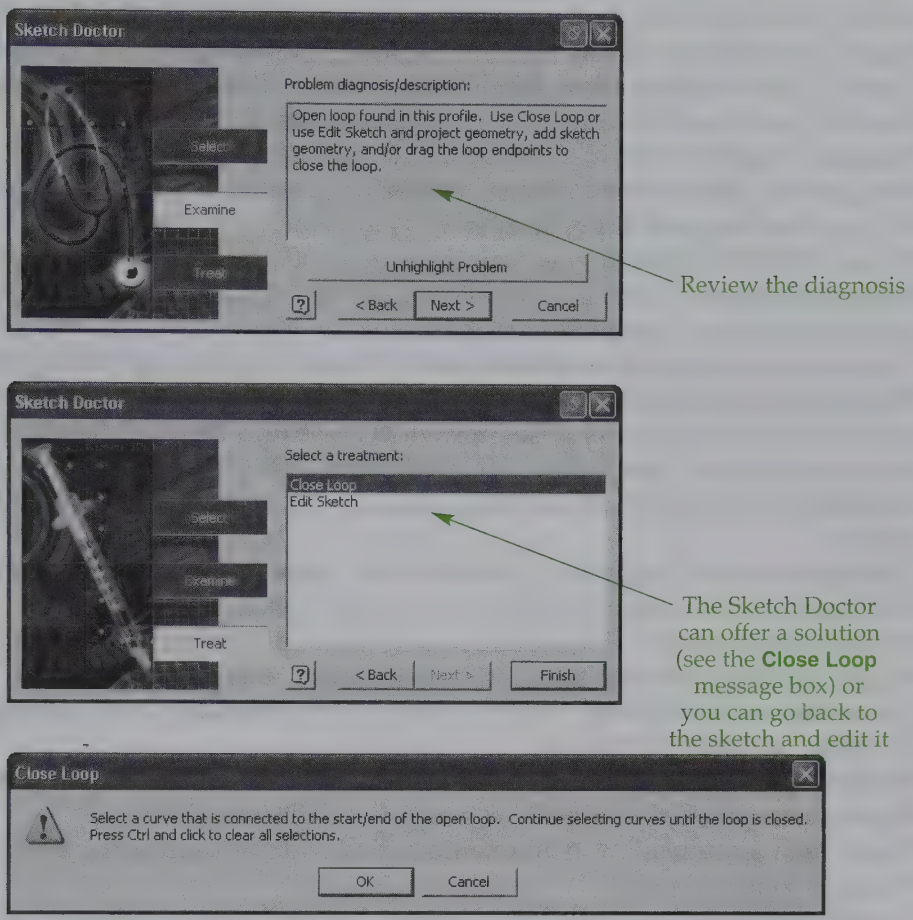
tests are selected. Once you determine which diagnostic tests you want to use, pick the **OK** button. The sketch icons and sketch problems are displayed in the **Select a problem to recover:** list box. Also notice your sketch and the sketch problems are highlighted.

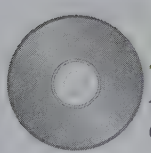
Pick the **Next** button to continue the **Sketch Doctor** application and display the **Examine** section of the **Sketch Doctor** dialog box. See Figure 4-53. The **Examine** section shows a description of the problem, and the diagnosis. You can also highlight or unhighlight the sketch if desired. Pick the **Next** button to access the **Treat** section of the **Sketch Doctor** dialog box. The treatment, or problem-solving, options are displayed in the **Select a treatment** list box. Choose the option you want to use to solve the problem and pick the **Finish** button. As shown in Figure 4-53, a solution is displayed in the graphics window. Follow the solution directions to solve the sketch problem.



Complete the Exercise on the Student CD.

Figure 4-53.
The Sketch Doctor will present a diagnosis and a solution.





Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. Identify what usually represents the first step in the creation of a model, and what they provide.
2. What is automatically selected, by default, when you start a new part model?
3. How do you know when you are in the sketch environment?
4. Give the basic function of sketching tools.
5. Why is it often helpful to project the center point, located in the **Origin** folder of the browser, before you sketch any geometry?
6. Give the two basic steps used to sketch a line.
7. List at least four options used to draw a line that references existing sketch geometry.
8. Identify the function of the **X,Y Coordinates** option.

9. Define spline.
10. Give the basic steps used to draw a center point circle.
11. When using the Ellipse tool, if you want to draw another ellipse, you pick a new center point, but what do you do if you do not want to draw another ellipse?
12. Identify the basic steps used to draw a three point arc.
13. Identify the steps that allow you to create arcs while using the Line tool.
14. Give the steps used to draw a two point rectangle.
15. What are fillets?
16. Provide the basic purpose of sketch points.
17. What is the basic use of hole centers?
18. Explain the difference between an inscribed and circumscribed polygon.
19. Discuss the basic function of the sketch Mirror tool, and specify what must be drawn before you can mirror an object.
20. Explain how spacing is measured, and give an example, when using the **Rectangular Pattern** tool.
21. What is the axis of rotation used for when creating a circular pattern and what is displayed when you choose a rotation axis?
22. Give the basic function of the Offset tool, and list at least four objects that can be offset.
23. Discuss the use of constraints, give another name for constraints, and identify how many constraints are used to fully define a sketch.
24. When using constraint tools that require you to pick two objects, give the relationship between the first and second object, and give an example.
25. Name the tool that allows you to display all the geometric constraints placed on an object or group of objects.
26. Explain how dimensions are used to constrain a sketch, and identify how many dimensions are needed to fully constrain or define an object.
27. Name the tool that allows you to automatically dimension and constrain several objects at once.
28. Provide the basic function of the Inventor Precise Input tool, including how use of the precise input tools affect constraints and the creation of constraints.
29. Discuss the function of sketch text in models and drawings and give at least two examples of use in each.
30. Define parameters, and briefly discuss how parameters relate to sketch geometry.

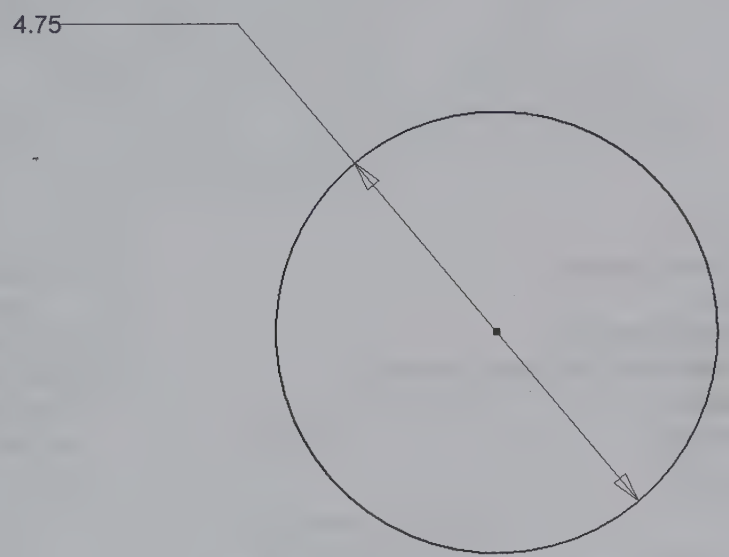
Problems

Instructions:

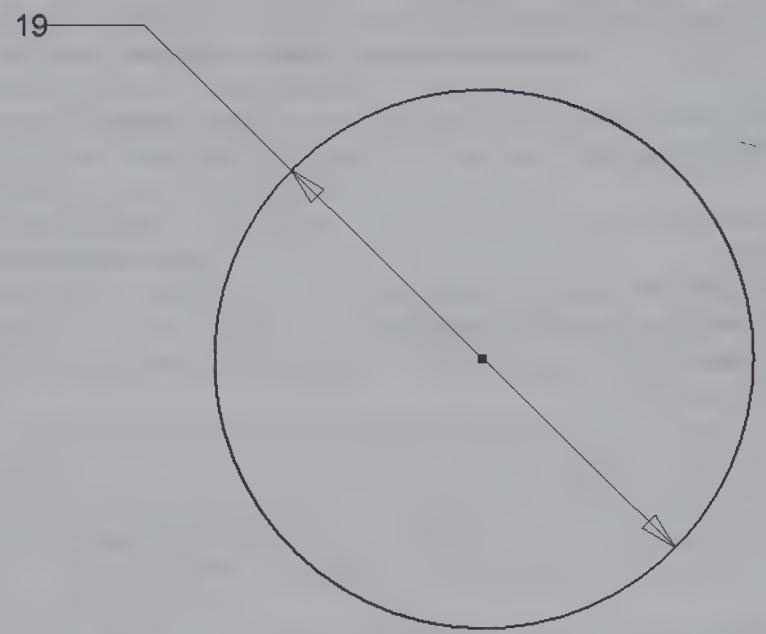
- Create sketches of the following objects using the techniques described in this chapter.
- Develop sketch geometry from the projected center point.
- Fully constrain each sketch.
- Use an **Equal** constraint for like objects not dimensioned in the problem figure.
- Add as much information as possible to the **Properties** dialog box.

Note: Sketch dimensions shown for reference are created using Autodesk Inventor and may not comply with ASME standards.

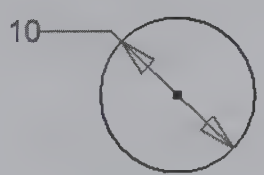
1. **Name:** C-Clamp Pin
   Units: Metric
   Template: PART-mm.ipt
   Material: Steel, High Strength Low Alloy
   Sketch plane: XY
   Save as: P4-1.ipt



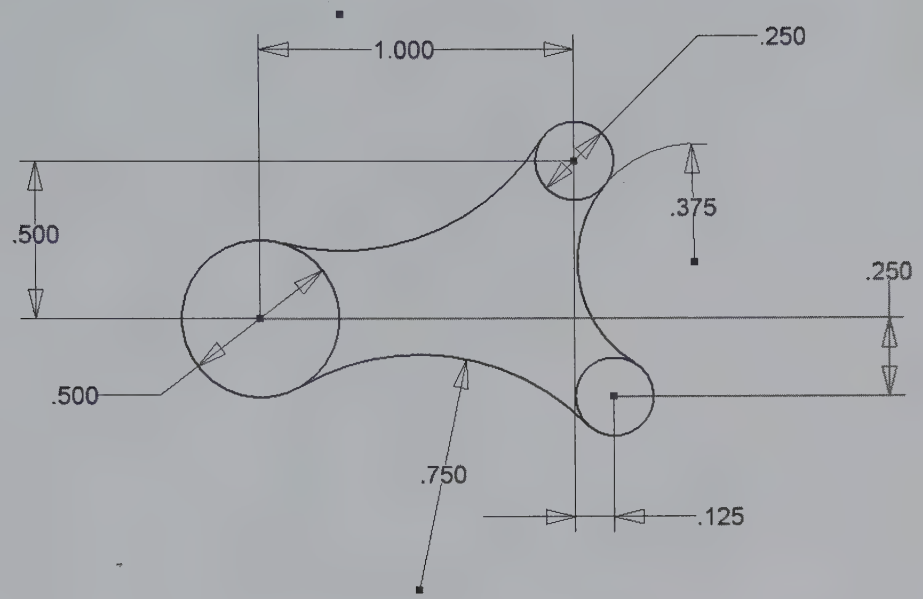
2. **Name:** C-Clamp Swivel
Units: Metric
Template: PART-mm.ipt
Material: Steel, High Strength Low Alloy
Sketch plane: XZ
Save as: P4-2.ipt



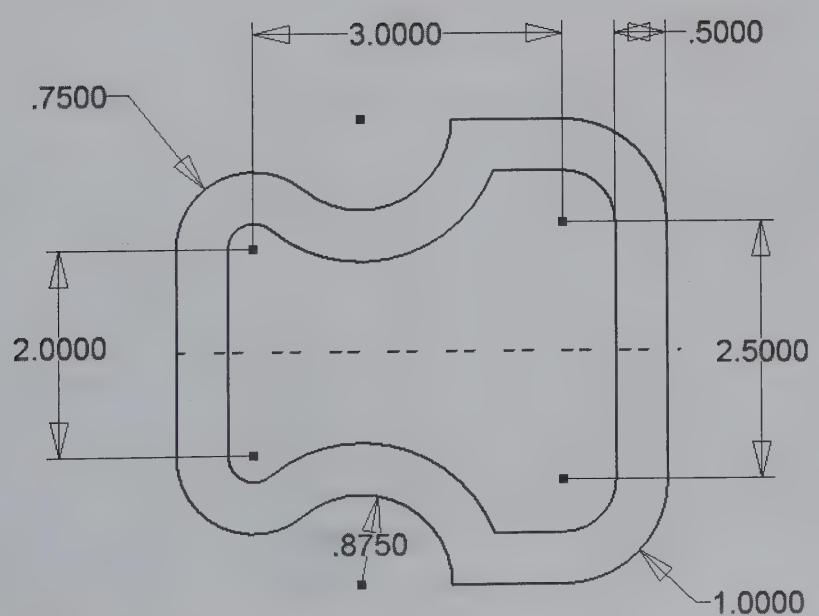
3. **Name:** C-Clamp Screw
Units: Metric
Template: PART-mm.ipt
Material: Steel, High Strength Low Alloy
Sketch plane: XZ
Save as: P4-3.ipt



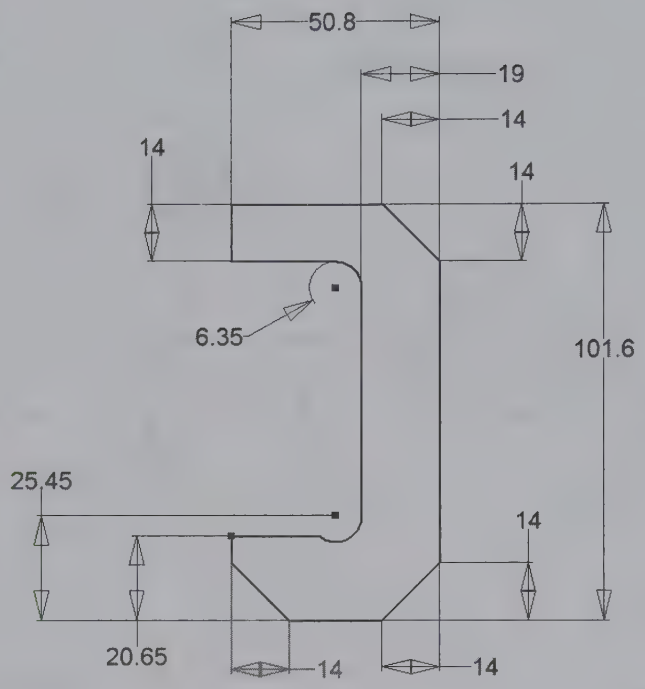
4. **Name:** Support Bracket
   Units: Inch
   Template: PART-in.ipt
   Material: Aluminum-6061
   Sketch plane: XY
   Save as: P4-4.ipt



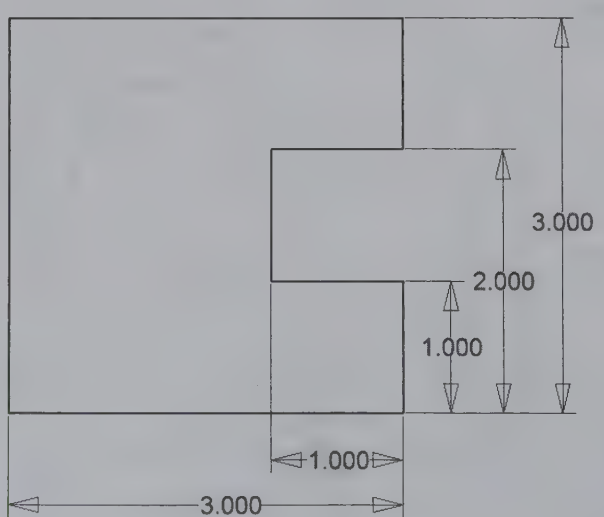
5. **Name:** Handle
   Units: Inch
   Template: PART-in.ipt
   Material: ABS Plastic
   Sketch plane: XY
   Save as: P4-5.ipt



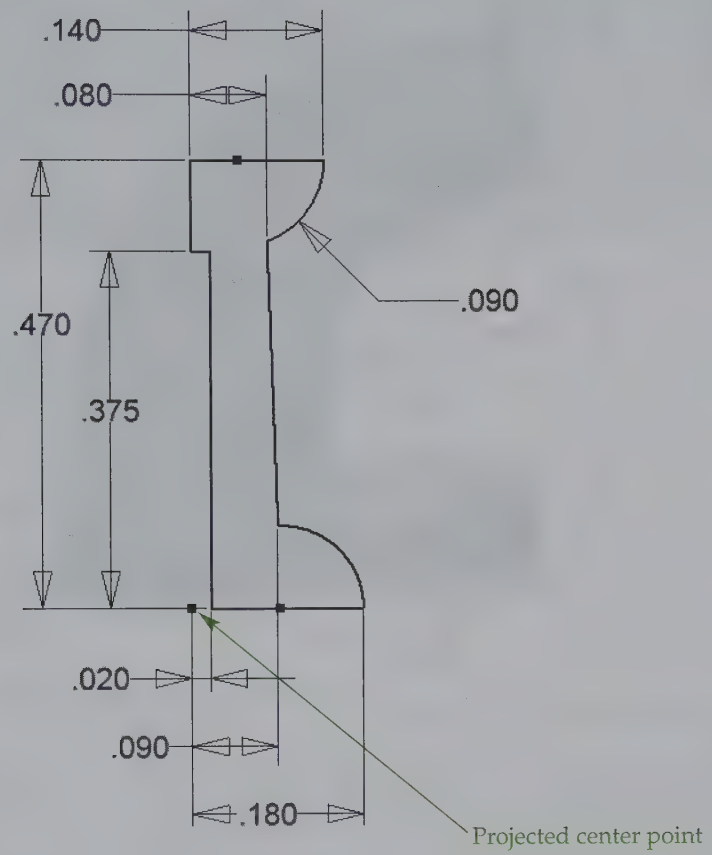
6. **Name:** C-Clamp Body
Units: Metric
Template: PART-mm.ipt
Material: Steel, High Strength Low Alloy
Sketch plane: XY
Save as: P4-6.ipt



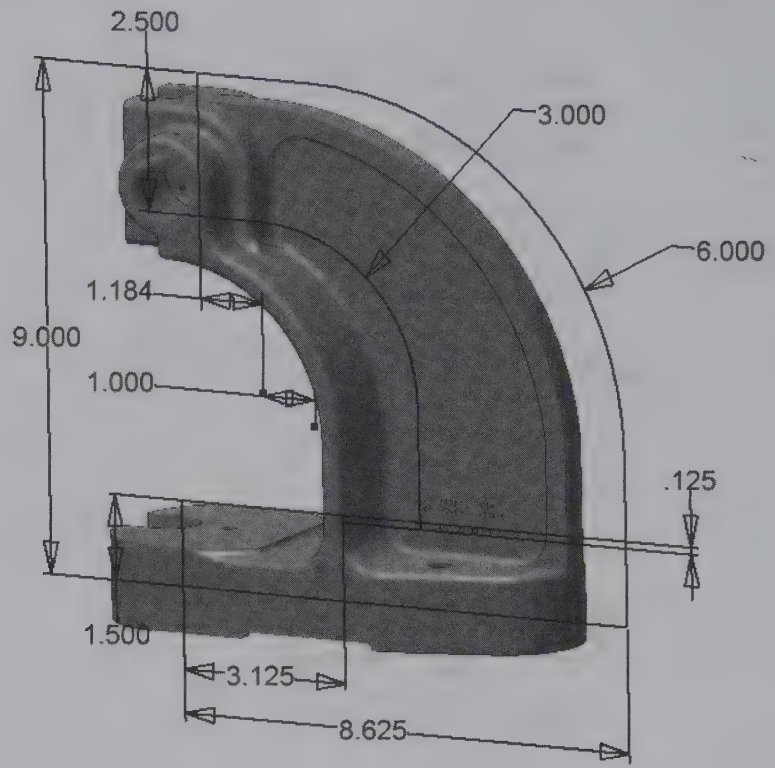
7. **Name:** Support Block
Units: Inch
Template: PART-in.ipt
Material: Default
Sketch plane: XZ
Save as: P4-7.ipt



8. **Name:** Push Pin Head
Units: Inch
Template: PART-in.ipt
Material: ABS Plastic
Sketch plane: XY
Save as: P4-8.ipt



Sketches are the foundation of even the most complicated models. This sketch is one of 36 that this model requires to create the basic shape, strengthening ribs, and precisely located holes.



CHAPTER 5

Basic Sketched Features

Learning Objectives

After completing this chapter, you will be able to do the following:

- Create extruded and revolved base features.
- Develop coils.
- Sketch on base features.
- Use automatic model edge referencing techniques.
- Generate additional extruded, revolved, and coil features.
- Use open sketch profiles.
- Add holes.
- Create ribs and webs.

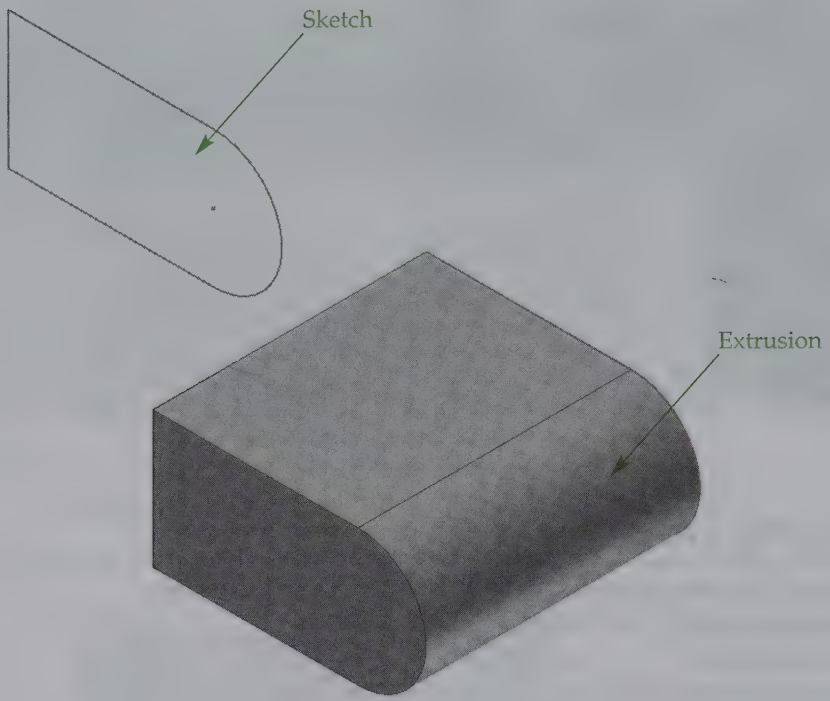
Once you create a sketch, you are ready to develop a three-dimensional part using Autodesk Inventor part feature tools. Usually, the first type of feature you create is a ***sketched feature***. As the name implies, sketched features are created from sketch geometry. Sketched features are typically simple and do not require very much time to develop. The initial feature, known as the ***base feature***, is often created from a sketch. Sketches and sketched features can be created anytime during part model development. Sketch features include extrusions, revolutions, sweeps, lofts, coils, holes, ribs, webs, embossments, and decals.

Creating Extruded Base Features

An ***extruded*** base feature is one of the most basic and often least complicated Inventor features. An example of an extrusion is shown in Figure 5-1. Access the **Extrude** tool by picking the **Extrude** button on the **Part Features** panel bar; picking the **Extrude** button on the **Part Features** toolbar; or pressing the [E] key.

The **Extrude** dialog box is displayed when you access the **Extrude** tool. See Figure 5-2. If only one sketch loop exists, as is often the case when creating a base feature, the sketch profile is already selected. To extrude a sketch with more than one profile available, pick the **Profile** button from the **Shape** area of the **Extrude** dialog box, if it is not already active. Then, pick the profile of the sketch you want to extrude. When the sketch profile is selected, a preview of the extrusion is displayed. See Figure 5-3.

Figure 5-1.
An example of an extrusion and the sketch used to create the extrusion.



NOTE

The extrusion shown in Figure 5-1 contains only one sketch profile. However, sketches can include many profiles. To create different features from a sketch with several profiles, use the **Share Sketch** option, covered later in this chapter.

When you create a complicated part, and multiple features exist, more options are available for developing extrusions. For the simple base feature in Figure 5-3, the following options are usable inside the **Extrude** dialog box:

- **Shape.** This tab, shown in Figure 5-2, controls primary extrusion shape characteristics and contains the following options:

Figure 5-2.
The **Extrude** dialog box, **Shape** tab, **Distance** termination option.

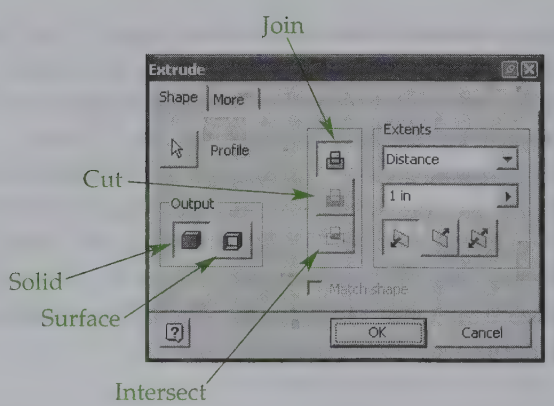
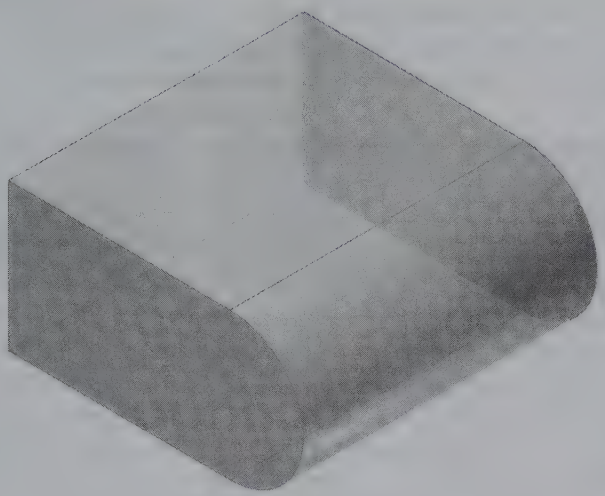


Figure 5-3.
An extrusion preview.



- **Output.** This area contains the **Solid** and **Surface** buttons. Pick the **Solid** button to generate a solid extrusion, as shown in Figure 5-1, or select the **Surface** button to create a surface extrusion, as shown in Figure 5-4. A *surface* is a volumeless shape that is primarily used for construction purposes, allowing you to generate models that are more advanced.
- **Operation.** These buttons allow you to choose whether the extrusion joins, cuts, or intersects another feature. However, when creating a base extrusion, only the **Join** button is available.
- **Extents.** This area contains options allowing you to specify the extrusion size and boundaries. See Figure 5-5. When creating a base feature with no work planes present, the **Distance** ending option is the only usable extent

Figure 5-4.
An example of a surface.

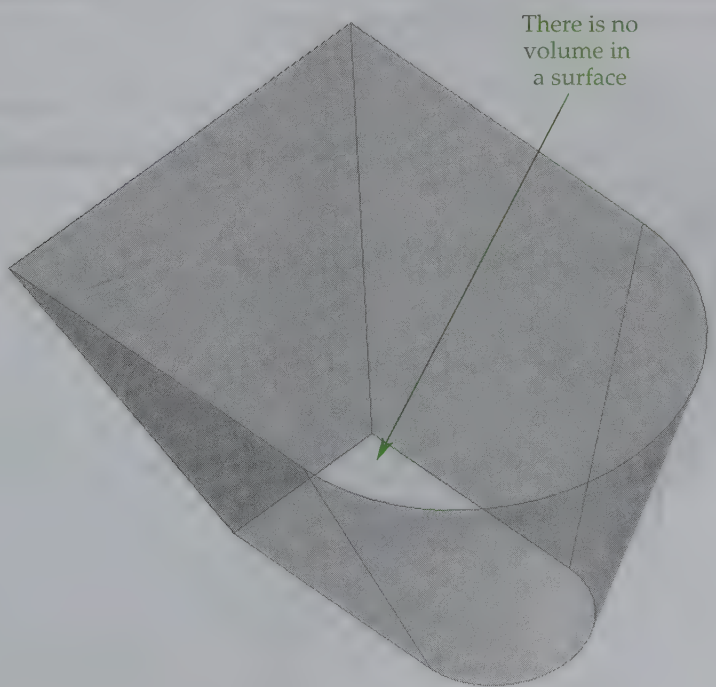
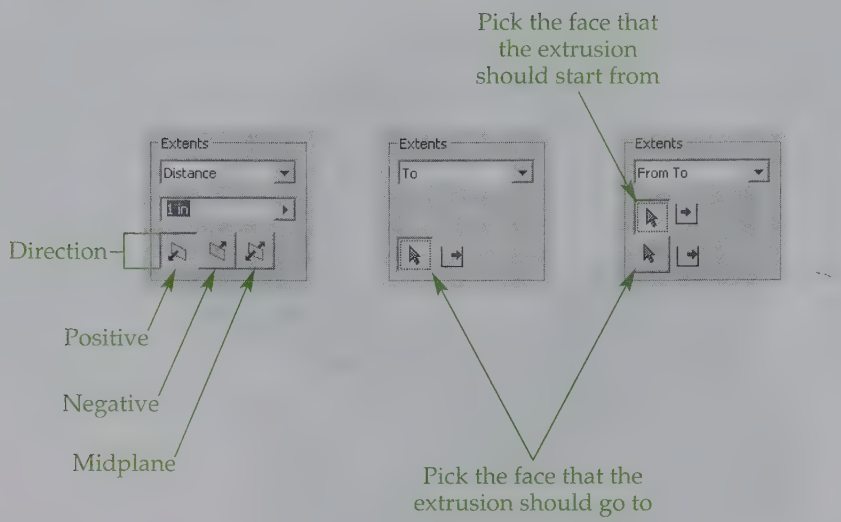
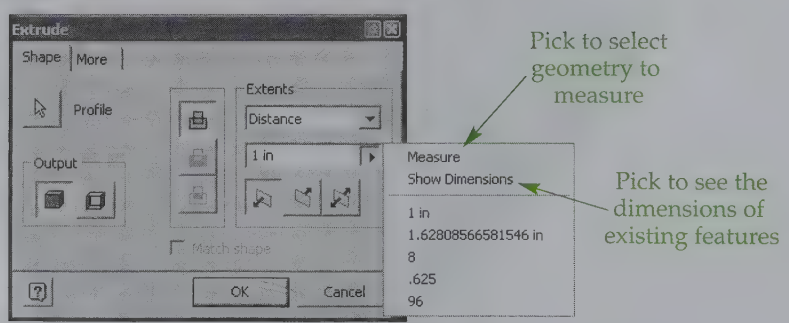


Figure 5-5.
The **Distance** termination.



parameter. Using **Distance** to end an extrusion allows the extrusion to be created along the sketch plane at a desired length. Specify a distance termination in the **Distance** edit box, or pick a distance from the list. The **To** and **From To** ending options allow you to use a face or work plane to control the beginning and end of the extrusion. You also have the option of picking **Measure** and **Show Dimensions**. Pick **Measure** if a sketch or feature edge is available. Pick **Show Dimensions** and a feature to see the dimensions of the feature. You can then pick one of the dimensions. See Figure 5-6. This allows you to reference an existing edge or parameter value to create an extrusion of the same length. You can also define the direction in which the extrusion occurs by selecting the **Positive**, **Negative**, or **Midplane** direction button. A positive direction extrudes the sketch in a positive direction from the sketch plane, a negative direction extrudes the sketch in a negative direction from the sketch plane, and a midplane direction divides the extrusion distance equally on both sides of the sketch plane.

Figure 5-6.
Using the **Measure** tool to reference an edge for the distance. Use the **Show Dimensions** tool to select a dimension from an existing feature.



PROFESSIONAL TIP



You can enter an extrusion distance using a parameter created in the **User Parameters** section of the **Parameters** dialog box by selecting the **List Parameters** option of the **Distance** cascading submenu and then selecting the desired parameter.

- **More.** This tab, shown in Figure 5-7, controls additional extrusion characteristics. The options in the **Alternative Solution** area are used when defining a **To** or **From To** extrusion extent type. However, you can taper any type of extrusion using the **Taper** edit box available in the **More** tab of the **Extrusion** dialog box. If you want to taper the extrusion, as shown in Figure 5-8, specify a taper angle in the **Taper** edit box, or pick one from the list. You can also use the **Measure** option. Pick the **Infer iMates** check box to add iMates to circular extrusion edges. iMates can be used to assemble components.

Figure 5-7.
The **Extrude** dialog box, **More** tab.

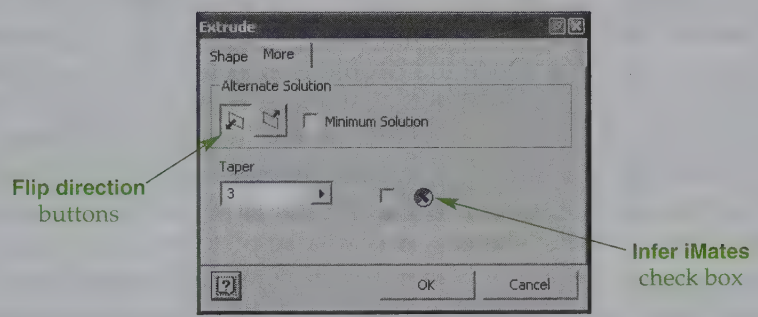
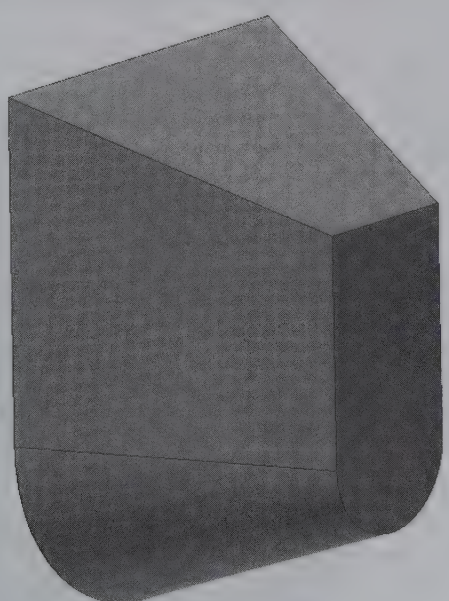


Figure 5-8.
Tapering an extrusion.



Once you specify all extrusion options, pick the **OK** button to complete the extruded feature.



Exercise 5-1 Complete the Exercise on the Student CD.

Creating Revolved Base Features

A ***revolved*** feature is very similar to an extrusion. Revolutions create a feature around an axis, in a circular path, instead of creating a feature in a linear direction, along a plane, like an extrusion. An example of a revolution is shown in Figure 5-9. Pick the **Revolve** button on the **Part Features** panel bar, pick the **Revolve** button on the **Part Features** toolbar, or press the [R] key to access the **Revolve** tool, and display the **Revolve** dialog box. See Figure 5-10. If only one sketch loop exists, as is often the case when creating base features, the sketch profile is already selected. To revolve a sketch with more than one profile available, pick the **Profile** button from the **Shape** area of the **Revolve** dialog box if it is not already active. Then, pick the profile of the sketch you want to revolve. Once the profile is selected, you must pick an axis. In order to select an axis you may need to pick the **Axis** button. In contrast to an extrusion, an axis is required to create a revolved feature because the axis defines where the revolution occurs, the line of symmetry, and the diameter of the revolved feature. Axes are created in the sketch environment using the **Line** tool or by projecting an axis from the **Origin** folder onto the sketch. You can also pick one of the axes from the **Origin** folder as the line of revolution without actually creating a sketch axis. When the sketch profile and the axis are selected, a preview of the extrusion is displayed. See Figure 5-11.

Figure 5-9.
An example of a revolved feature and the sketch used to create the revolved feature.

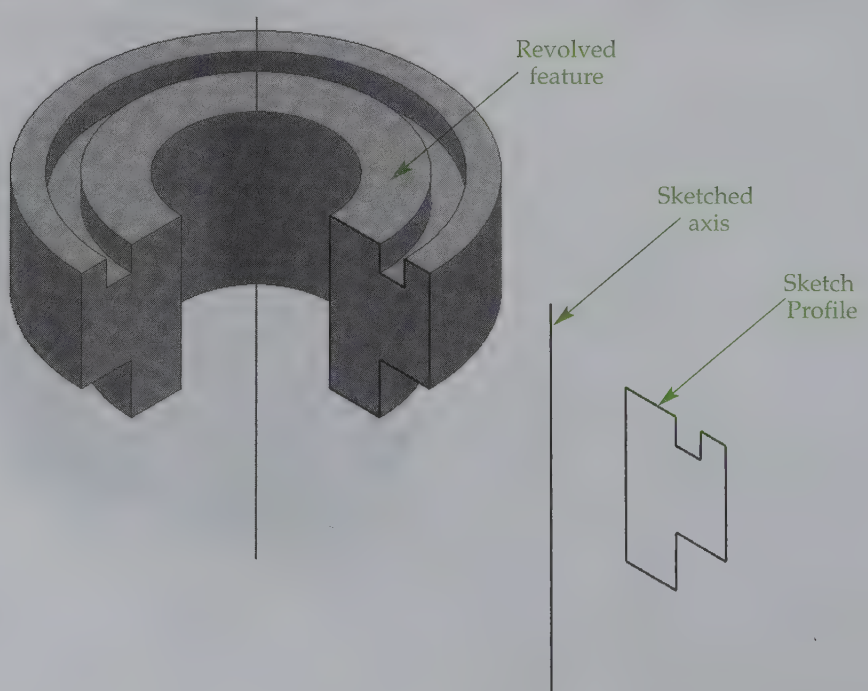
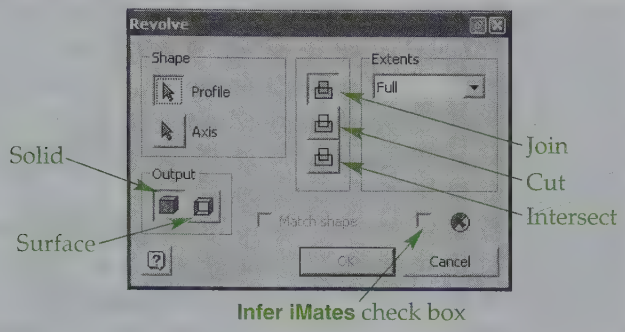


Figure 5-10.
The **Revolve** dialog box, **Full** rotation option.



PROFESSIONAL TIP

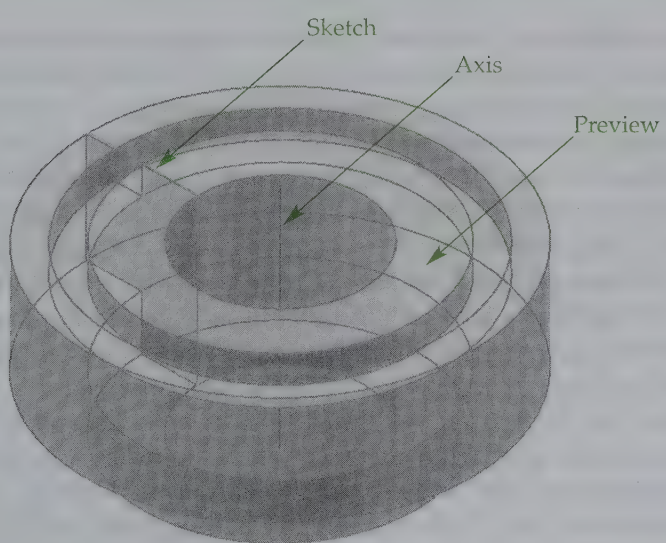


Project the center point and create a constraint between the origin and sketch geometry. This allows the sketch profile to become fully constrained to the origin, not just a point in space. It also allows you to define the diameter of the revolved feature.

When you create a more complicated part, and multiple features exist, more options are available for developing revolved features. For a simple base feature, the following options are usable inside the **Revolve** dialog box:

- **Output.** This area contains the **Solid** and **Surface** buttons. Pick the **Solid** button to generate a solid revolution, or select the **Surface** button to create a volumeless, construction surface revolution.
- **Operation.** Only the **Join** button is available for a base feature. When you create additional features all operation buttons are offered.

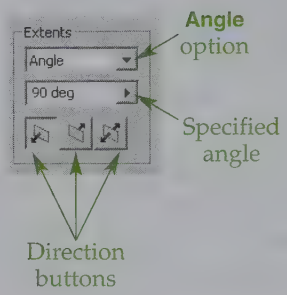
Figure 5-11.
A revolution preview.



- **Extents.** This area contains options allowing you to specify the angle or amount of revolution that occurs. The **Full** option is active by default. See Figure 5-10. The **Full** option allows you to revolve a feature 360°, or completely around the axis. If you do not want to make a full revolution, as shown in Figure 5-9, pick the **Angle** option from the drop-down list. See Figure 5-12. Using the **Angle** termination option allows you to specify a revolution angle of greater than 0° and less than 360°. Select an angle from the list or enter a value in the **Angle** edit box. You can also pick **Measure**. This allows you to reference an existing edge and revolve around the measurement angle.

Once you specify the angle you want to use to revolve the feature, pick the desired direction button. The direction buttons are the same as those used in the **Extrude** tool and control the side of the sketch the revolution occurs if you use the **Angle** option. Pick the **Infer iMates** check box to add iMates to circular revolution edges. iMates can be used to assemble components. After you specify all revolve options, pick the **OK** button to complete the revolved feature.

Figure 5-12.
The **Angle** option of the **Extents** area.





Complete the Exercise on the Student CD.

Developing a Coil

A coil is similar to a revolved feature, but the sketch profile travels along the length of the axis while rotating around the axis. *Coils* are spiral, or helix, features primarily used to create springs and detailed threads. See Figure 5-13.

The first step in creating a coil feature is to generate a sketch profile. When creating detailed threads, the sketch is typically placed on a work plane. The sketch profile is then cut into an existing feature, such as an extruded cylinder. Once you develop a sketch profile, as shown in Figure 5-14, you are ready to create a coil. Pick the **Coil** button on the **Part Features** panel bar, or pick the **Coil** button on the **Part Features** toolbar to access the **Coil** tool and display the **Coil** dialog box. See Figure 5-15. The **Coil Shape** tab of the **Coil** dialog box is initially displayed and contains a **Shape** and **Rotation** area. The following buttons are available inside the **Coil Shape** tab:

- **Profile.** This button allows you to select a sketch profile. If only one sketch profile exists, as is often the case when creating a base feature, the sketch is already selected. To choose a sketch with more than one profile available, pick the **Profile** button if it is not already active. Then pick the sketch profile you want to coil.

Figure 5-13.
An example of a spring and threads created using the **Coil** tool.



PROFESSIONAL TIP



Project the center point and create a constraint between the origin and sketch geometry. This allows the sketch profile to become fully constrained to the origin, not just a point in space. It also allows you to define the diameter of the coil and effectively reference the default axes from the **Origin** folder.

- **Axis.** This button allows you to select an axis, and is typically automatically available when you select the profile. If not, you must pick the **Axis** button in order to select an axis. An axis is required to create a coil, because the axis defines the direction and diameter of the coil. Axes are created in the sketch environment using the **Line** tool, or by projecting an axis from the **Origin** folder onto the sketch. You can also pick one of the axes from the **Origin** folder without actually creating a sketch axis. When the sketch profile and the axis are selected, a preview of the coil is displayed. See Figure 5-16.
- **Flip.** This button allows you to redefine the direction of the coil.
- **Rotation.** Specify the coil's direction of rotation as clockwise or counterclockwise.

Figure 5-14.
An example of a sketch used to create a spring and a sketch used to create threads.

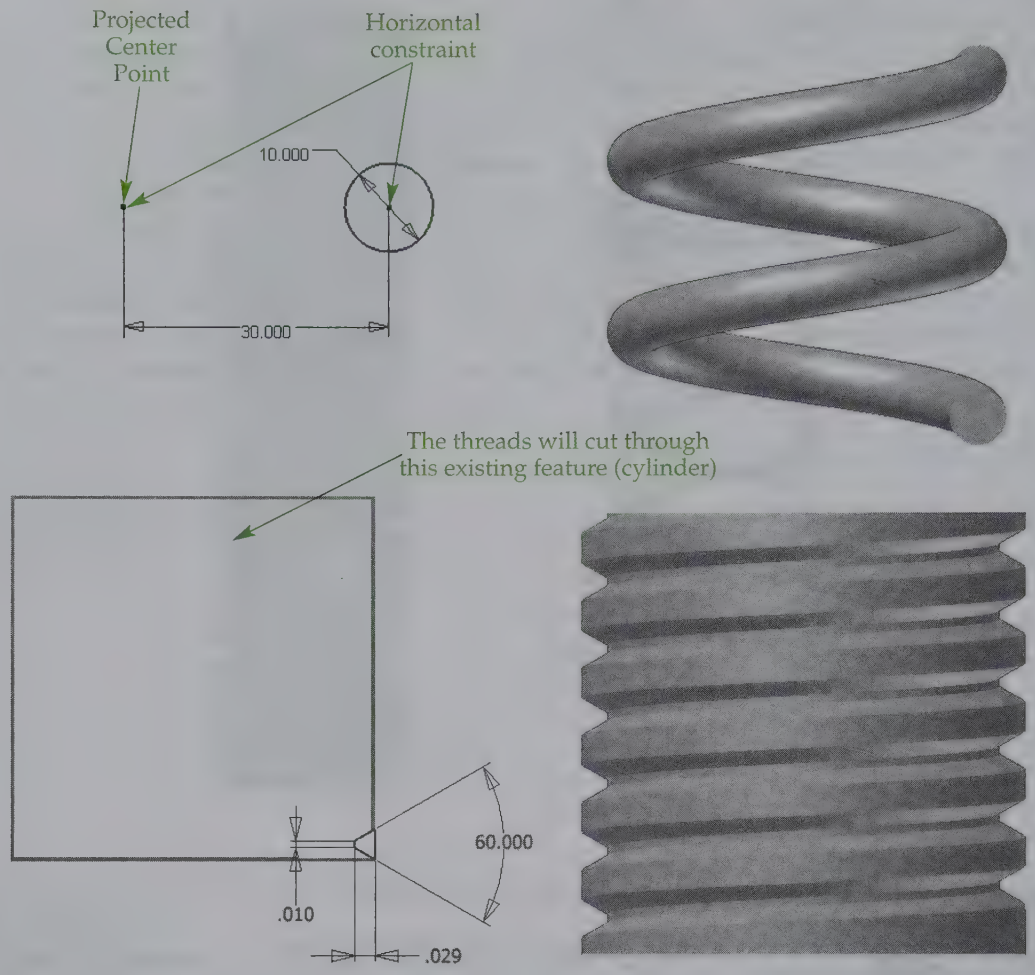


Figure 5-15.
The **Coil** dialog box, **Coil Shape** tab.

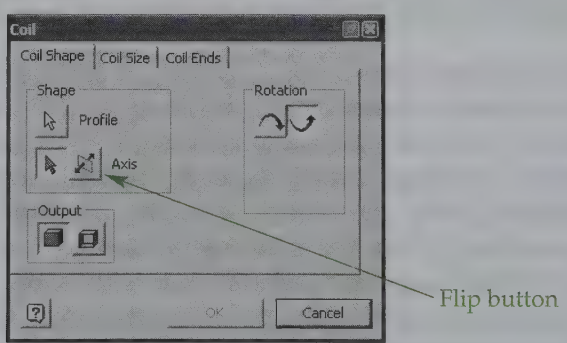
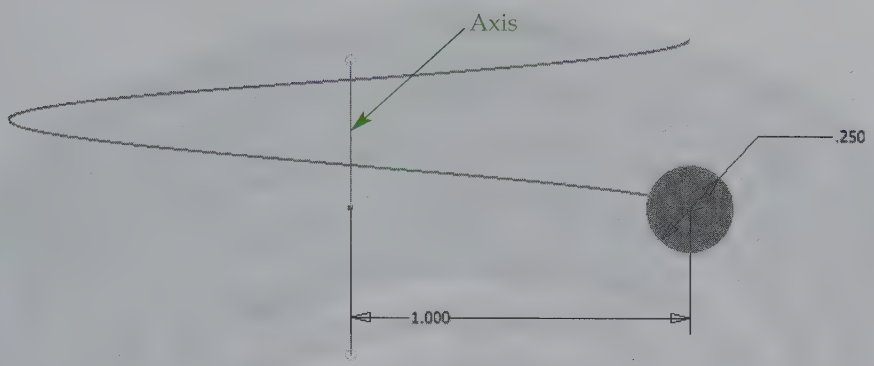


Figure 5-16.
A coiled feature preview.



NOTE

A counterclockwise rotation corresponds to right-hand threads and a clockwise rotation corresponds to left-hand threads. The **Join**, **Cut**, and **Intersect** operation buttons are only available when you create a coil on an existing feature, such as an extruded cylinder.

Once you fully define the coil shape, pick the **Coil Size** tab of the **Coil** dialog box to specify the size of the coil. See Figure 5-17. Then pick one of the following options from the **Type** drop-down list according to the coil information that you want to input:

- **Pitch and Revolution.** This option allows you to define the coil height based on pitch and revolution values. *Pitch* is the distance parallel to the axis between a point on one coil spiral to the corresponding point on the next coil spiral. *Revolution* is the number of spirals, or 360° loops, that the coil creates. Selecting the **Pitch and Revolution** option activates the **Pitch** and **Revolution** edit boxes. Pick the arrows and select a pitch and revolution, or enter the desired values. In addition to the **Pitch** and **Revolution** edit boxes, the **Taper** edit box is available, which allows you to specify a taper angle for the coil. See Figure 5-18. Pick the arrow and select an angle, or enter the desired value.

Figure 5-17.
The **Coil** dialog box, **Coil Size** tab.

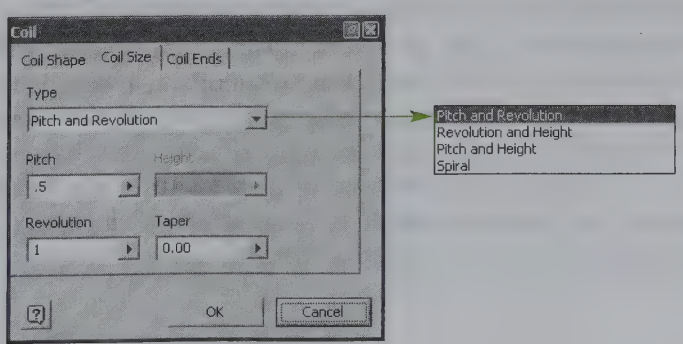
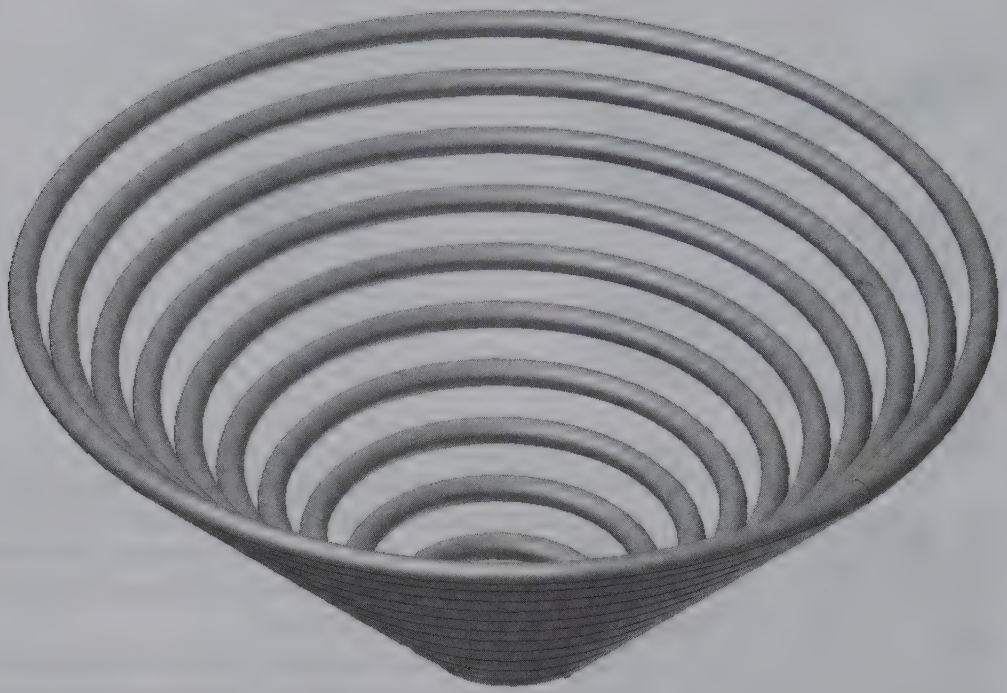


Figure 5-18.
An example of a tapered coil (45° taper).



- **Revolution and Height.** This choice allows you to define the coil pitch based on revolution and height values. ***Height*** is the total depth of the coil from the center of the starting profile to the center of the ending profile. Selecting the **Revolution and Height** option activates the **Revolution** and **Height** edit boxes. Pick the arrows and select a revolution and height, or enter the desired values. The **Taper** edit box is also available.
- **Pitch and Height.** This choice allows you to define the coil revolution based on pitch and height values. Selecting the **Pitch and Height** option activates the **Pitch** and **Height** edit boxes. Pick the arrows and select a pitch and height, or enter the desired values. The **Taper** edit box is also available.
- **Spiral.** This choice allows you to create a totally different type of coil shape, without any height. See Figure 5-19. The coil size is defined using pitch and revolution input values, and no height is calculated. Selecting the **Spiral** option activates the **Pitch** and **Revolution** edit boxes. Pick the arrows and select a pitch and revolution, or enter the desired values. The **Taper** edit box is not available.

Once you define the coil size, pick the **Coil Ends** tab of the **Coil** dialog box to specify the ends of the coil. See Figure 5-20. You can define the form of the coil ends for the start and end of the coil as natural or flat by selecting the desired option from the drop-down list. A ***natural end*** is the result of the pitch, revolution, height, and profile of the coil. A natural end does not allow you to define a transition or flat angle, because these angles are 0°. A natural end is considered unaltered or untreated. A ***flat end*** has adjustments of the last coil to create a flat end of the spring. Compare the Start and End coil ends of Coil1 in Figure 5-21. Selecting the **Flat** end option activates the **Transition Angle** and **Flat Angle** edit boxes. The transition angle is the number of degrees a coil end travels, or transitions, with pitch. Figure 5-21 shows the difference between a natural end and

Figure 5-19.
An example of a spiral coil.

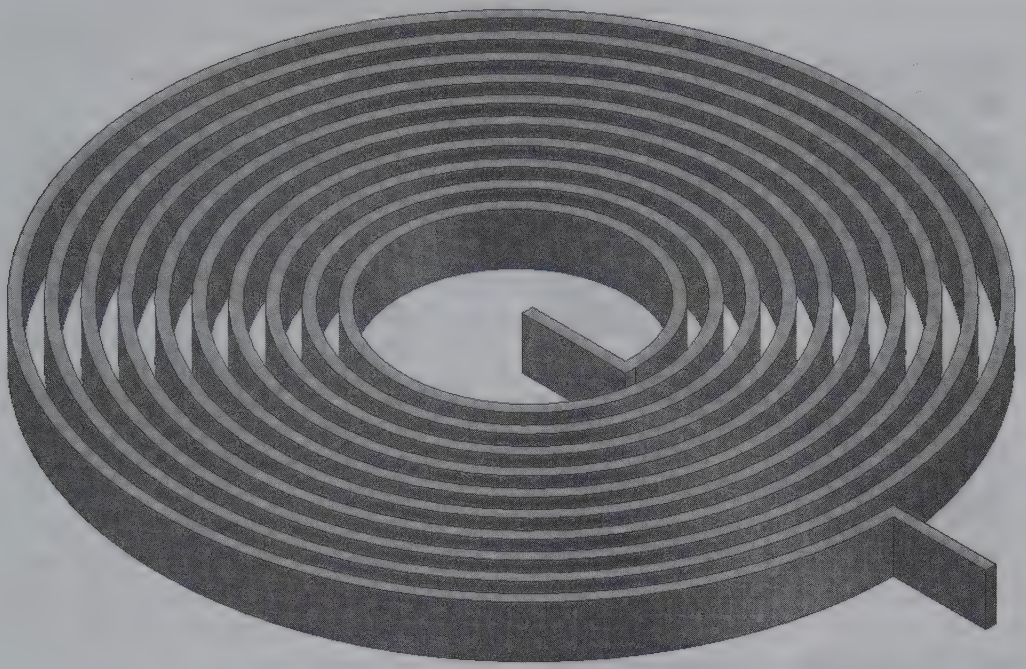
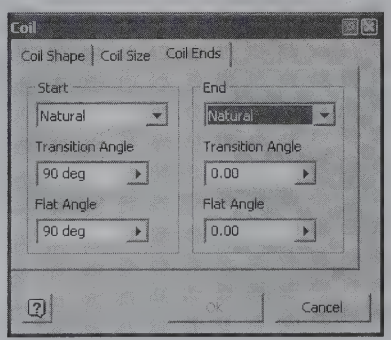


Figure 5-20.
The **Coil** dialog box, **Coil Ends** tab.



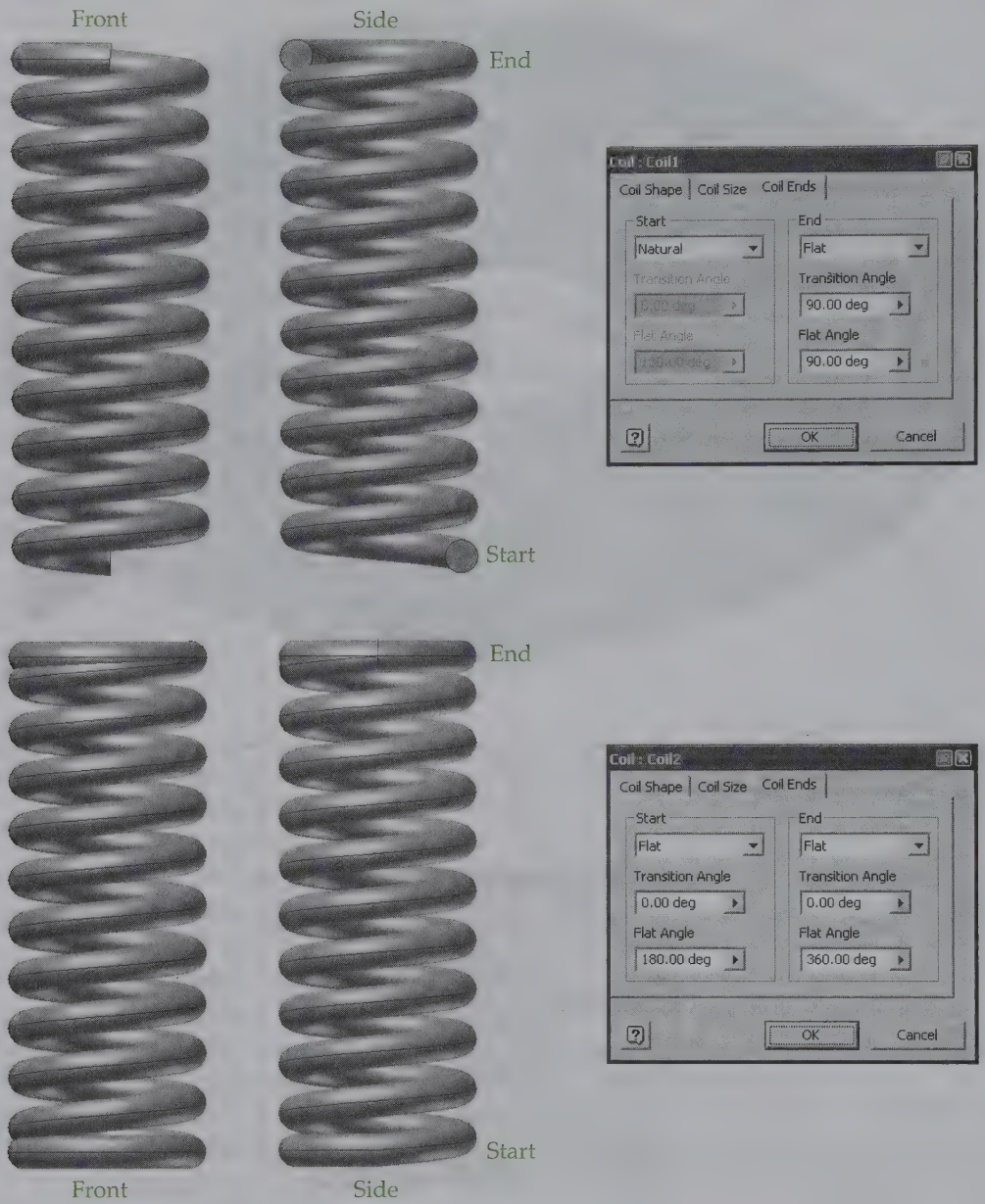
an end with a 90° transition angle and 90° flat angle. The flat angle is the number of degrees a coil end travels without pitch. Coil2 in **Figure 5-21** shows a transition angle of 0° and a flat end of 180°. Also shown is an end with a transition and flat end. When you complete the coil configuration, pick the **OK** button to create the coiled feature.

NOTE



Applying transition and flat angles to a coil does not change the profile geometry.

Figure 5-21.
Examples of transition and flat angles applied to coil ends.



PROFESSIONAL TIP



To create realistic, detailed threads, you may want to use the **Coil** tool. However, for most applications, the **Thread** tool is adequate for generating thread representations.



Exercise 5-3

Complete the Exercise on the Student CD.

Sketching on a Base Feature

Once you create a base feature using a sketched feature tool such as **Extrude**, **Revolve**, or **Coil**, you can further develop a part by positioning placed features such as shells, threads, fillets, chamfers, and face drafts, thickening a face or offsetting a surface, inserting catalog features, patterning features, or adding work features for sketching additional profiles or paths to use with sketched feature tools such as **Loft** and **Sweep**. An alternative to these techniques is to create a sketch on or around the existing base feature in order to create more sketched features. For example, you can generate an extruded base feature and then cut another extrusion through the base feature. Additional sketched features can be developed using one of the following techniques:

- Extrude, revolve, coil, or emboss a sketch profile to develop a feature that joins, cuts, or intersects the base feature.
- Add a decal to an existing feature by importing a .bmp file into a sketch.
- Create a hole in the base feature.
- Sketch lines to make ribs and webs.

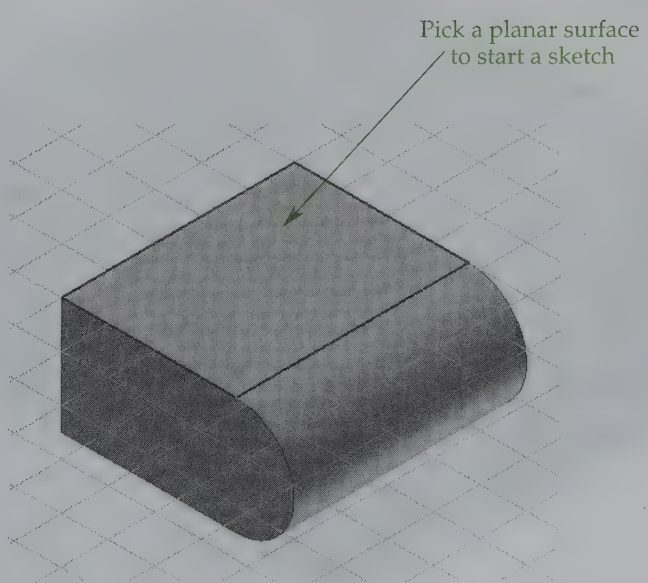
To create a sketch on a base feature or any existing feature, select the face on which you want to open a sketch, and pick the **Sketch** button on the **Inventor Standard** toolbar; right-click on the face on which you want to open a sketch, and pick the **New Sketch** pop-up menu option; or use the [S] hot key, and pick the face on which you want to sketch; or pick the face and press the [S] key. Figure 5-22 shows a new sketch plane opened on an extruded base feature. When you open a new sketch on an existing feature, you are in the sketch work environment. All the tools and applications described in the sketching chapter apply.

PROFESSIONAL TIP



You can also create a new sketch on any of the reference planes located in the **Origin** folder, including the **YZ Plane**, **XZ Plane**, and **XY Plane**.

Figure 5-22.
An example of a sketch plane opened on a feature face.



NOTE

The techniques and applications used to open a sketch on a base feature apply to any feature face, or plane.

Automatic Model Edge Referencing

Autodesk Inventor contains tools that allow you to use existing part edges for the creation of new sketch geometry. Two of these tools are found in the **Sketch** tab of the **Options** dialog box, located in the **Tools** pull-down menu. Picking the **Autoproject edges during curve creation** check box allows you to project the edge of an existing feature onto the sketch plane while using sketch tools. See Figure 5-23. To use the **Autoproject edges during curve creation** tool, you must first access a sketch tool such as **Line**, **Spline**, **Circle**, or **Arc**. Then move your cursor over the edge you want to project. The edge should change color to signify that it is projected. If you want to project another edge, repeat the process. Edges projected using the **Autoproject edges during curve creation** tool are available for reference only and do not become actual sketch geometry unless you constrain new objects to the projection. This means that if you do not actually pick a location on a projected edge when you create new sketch objects, the projection is eliminated when you exit the sketch, and is not available for feature development.

Selecting the **Autoproject edges for sketch creation and edit** check box allows Autodesk Inventor to automatically project all the selected face edges onto the new sketch plane when you open a new sketch. See Figure 5-24. You know that the edges are projected when they change color and are constrained. Edges projected using the **Autoproject edges for sketch creation and edit** tool automatically become sketch geometry. Use this technique to almost instantly produce a new sketch profile based on the outline of a feature face.

Figure 5-23.
An example of an autoprojected feature edge.

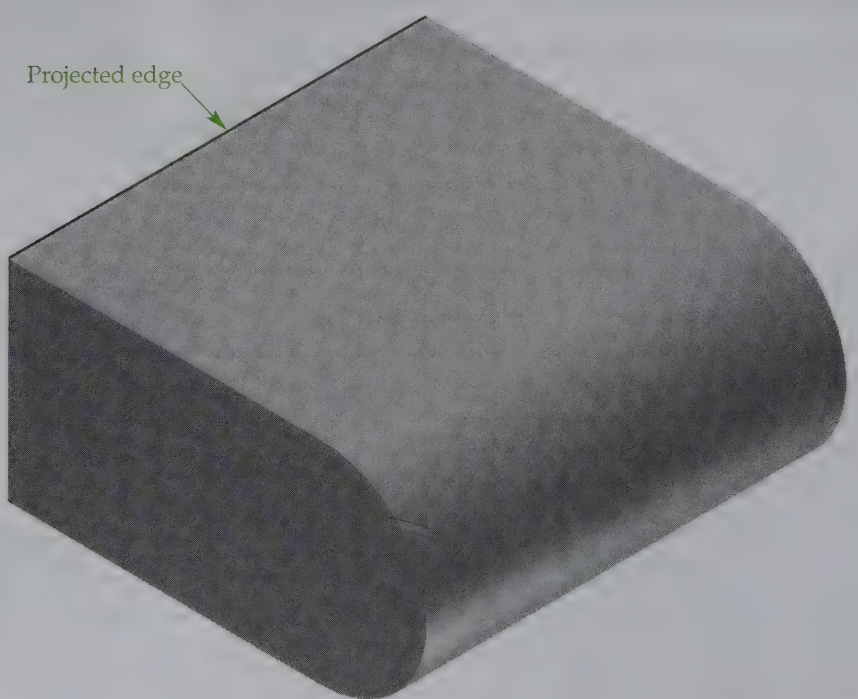
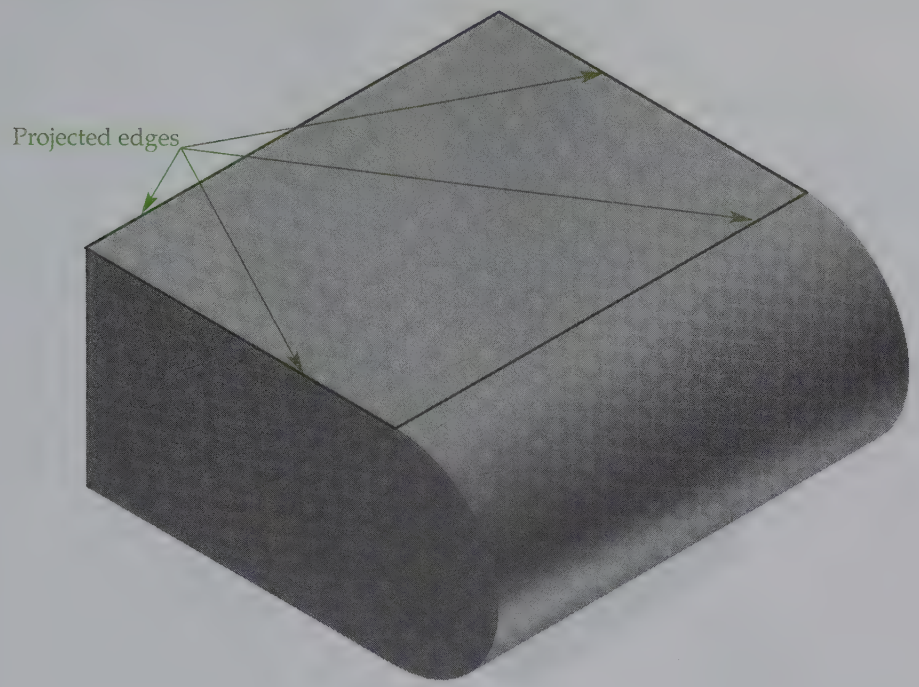


Figure 5-24.
An autoprojection of feature edges.



NOTE

You can constrain and dimension your sketch to feature edges without projecting the edges. Constraining and adding dimensions automatically projects the referenced edges.

Exercise 5-4 Complete the Exercise on the Student CD.

Exercise 5-5 Complete the Exercise on the Student CD.

Creating Additional Extrusions

Now that you have created a base feature for your part and opened a sketch on an existing feature, you are ready to create additional extrusions. See Figure 5-25. If you access the **Extrude** tool when one or more features exist, the **Extrude** dialog box contains additional usable options, because reference faces are present. See Figure 5-26. The following preferences are available inside the **Extrude** dialog box:

Figure 5-25.
An example of adding an extruded feature to a base feature.

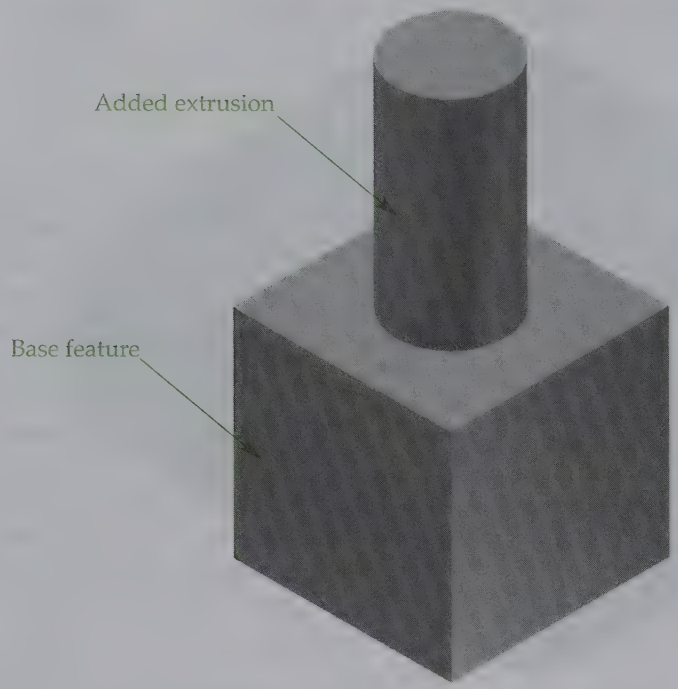
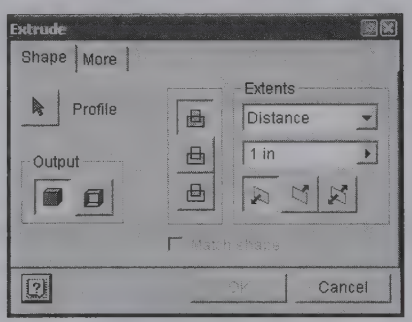


Figure 5-26.
The **Extrude** dialog box with all options available.



- **Operation**. In addition to the **Join** button, the **Cut** and **Intersect** buttons are available when creating more features. Picking the **Cut** button allows you to remove material from an existing feature by cutting through the existing feature with a sketch profile. See Figure 5-27. Picking the **Intersect** button allows you to create a feature by using existing intersecting features. When you apply an intersect operation to two or more features, the intersecting portion of the feature or features is retained, and all other material is removed. For example, Figure 5-28A shows an existing feature to which a sketch has been added, the resulting intersection extrusion is shown in Figure 5-28B.
- **Extents.** As shown in Figure 5-29, the following extrusion termination options are now usable inside the **Extents** area:
  - **To Next.** Selecting this option enables you to extrude a feature to the next possible face or plane. See Figure 5-30. To extrude a feature to the next face or plane, pick the sketch profile, and select the direction you want the extrusion to occur. Then pick the **OK** button to create the extruded feature.

Figure 5-27.
An example of a cut extrusion.

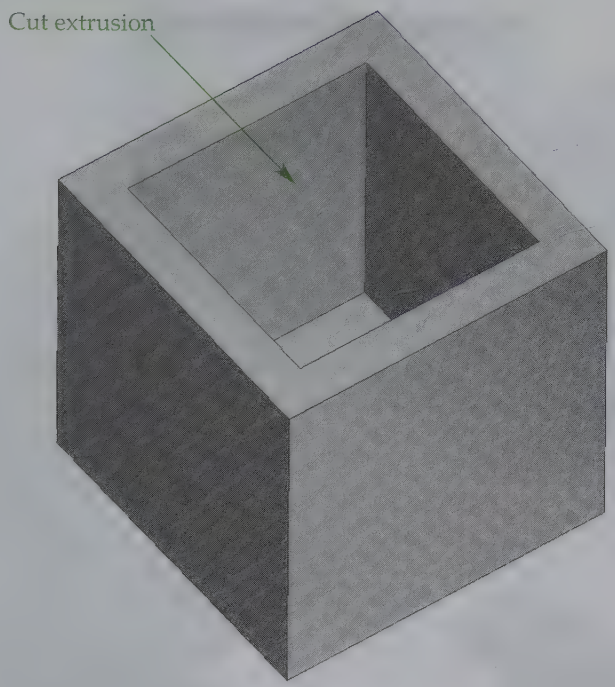


Figure 5-28.
An example of an intersection extrusion.

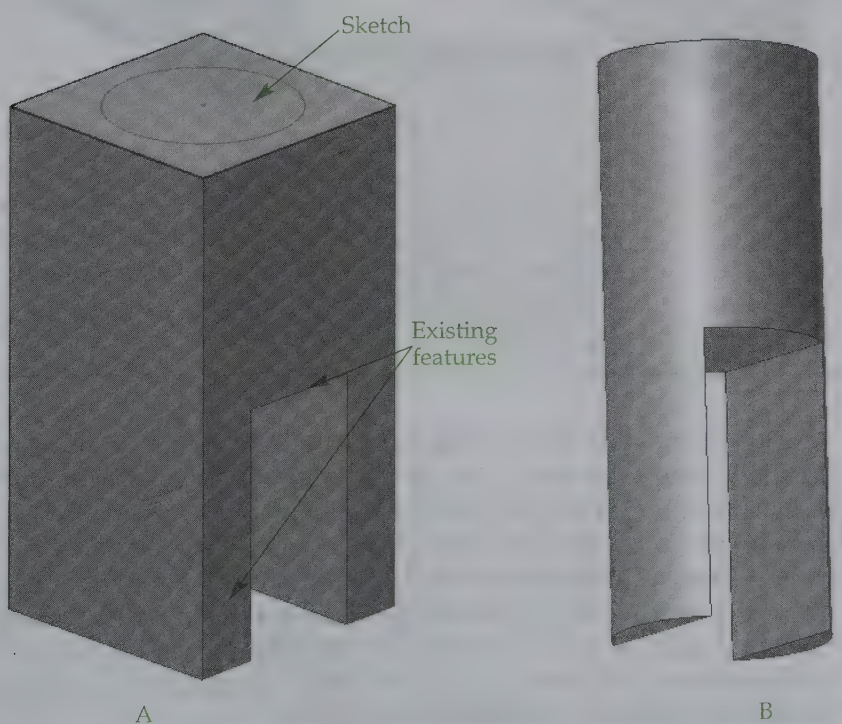


Figure 5-29.
The **Extents** extrusion termination options of the **Extents** area, for parts with multiple features.

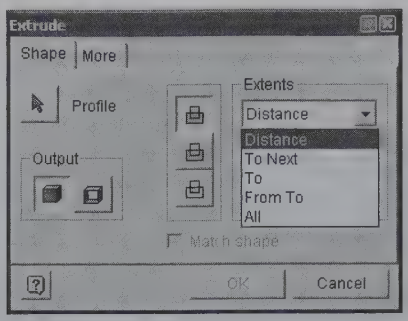
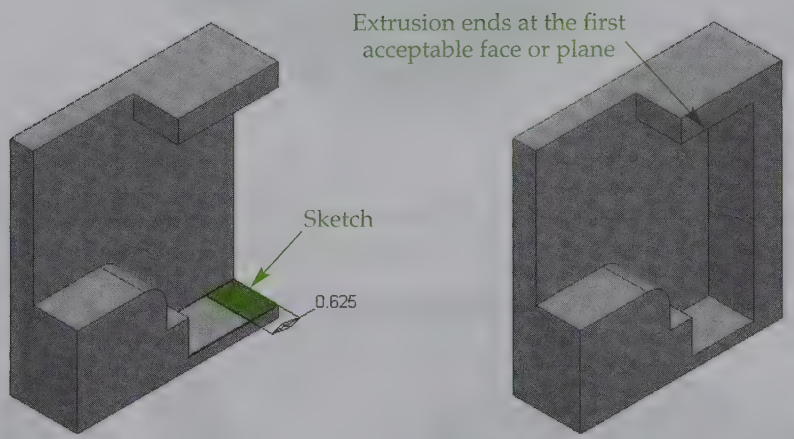


Figure 5-30.
Creating an extrusion using the **To Next** termination option.



NOTE

There is no equal side or midplane direction option available when extruding **To Next** face or plane.

If a face or plane does not exist that can end the extrusion, the feature cannot be created.

- **To.** This termination option is similar to the **To Next** option. However, selecting **To** allows you to choose a face or plane to end the extrusion, instead of the extrusion ending at the next possible face or plane. See Figure 5-31. To terminate an extrusion at a specified face or plane using the **To** option, first pick the sketch profile. Then pick the face or plane to which you want to extrude the feature. You may need to choose the **Select surface to end the feature creation** button if it is not already activated. See Figure 5-32. If the face or plane you choose does not intersect the extrusion path, as shown in Figure 5-31, you need to check the **Terminate feature on extended face** check box.

  If you have difficulty using the **To** extent option, you may need to specify additional extent information in the **Alternate Solution** area of the **More** tab. Use the **Flip Direction** buttons to change the direction of the extrusion. You can also select the **Minimum Solution** check box to terminate the extrusion at the nearest plane or face instead of possibly cutting through multiple planes or faces.

Figure 5-31.
Creating an extrusion using the **To** termination option.

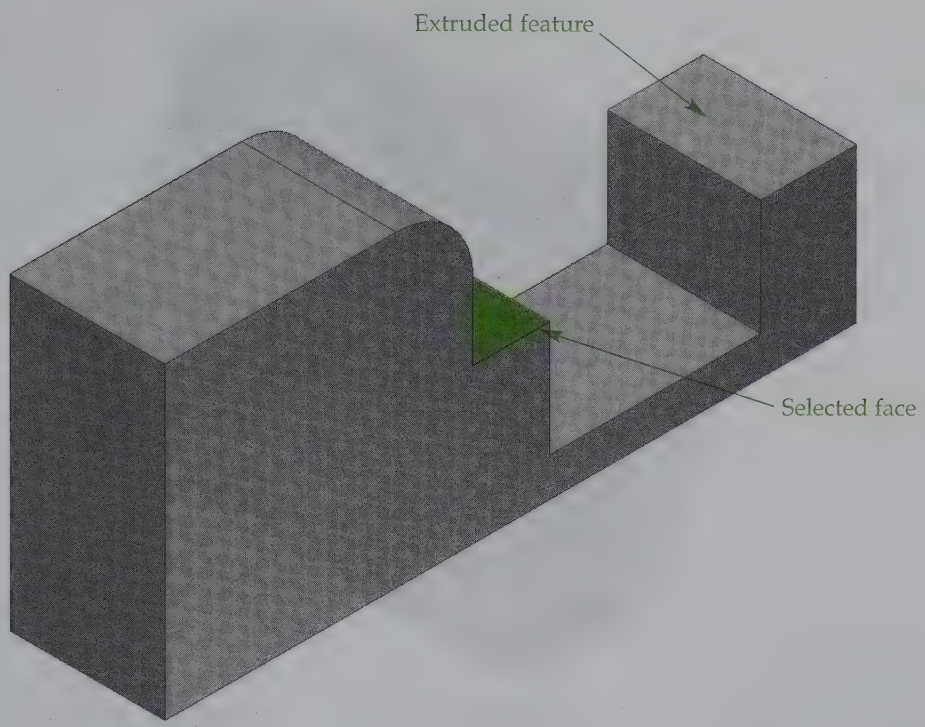
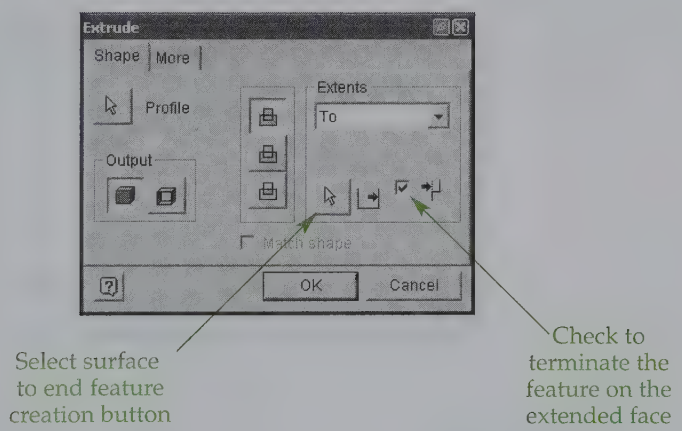


Figure 5-32.
The **To** option of the **Extrude** dialog box, **Extents** area.



- **From To.** This termination option is similar to the **To** option, but selecting **From To** allows you to choose a face or plane to begin and end the extrusion, instead of just the ending surface. See Figure 5-33. To begin and terminate an extrusion at a specified surface using this option, first pick the sketch profile. Then pick the face or plane you want to begin the extrusion from. You may need to choose the **Select surface to start the feature creation** button if it is not already activated. See Figure 5-34. Next, pick the face or plane you want to end the extrusion. You may need to choose the **Select surface to end the feature creation** button if it is not already activated. If the surfaces you choose to start and end the extrusion do not intersect the extrusion path, you need to check the **Terminate feature on extended face** check boxes.

Figure 5-33.
Creating an extrusion using the **From To** extents option.

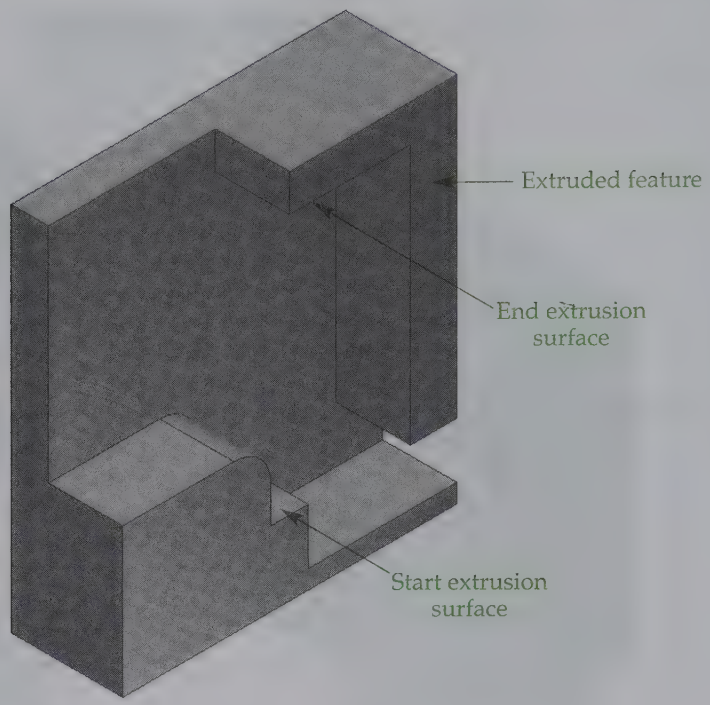
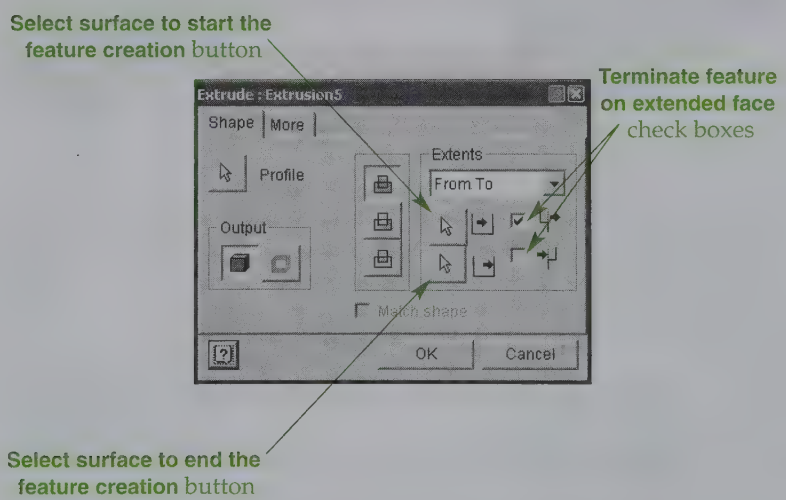


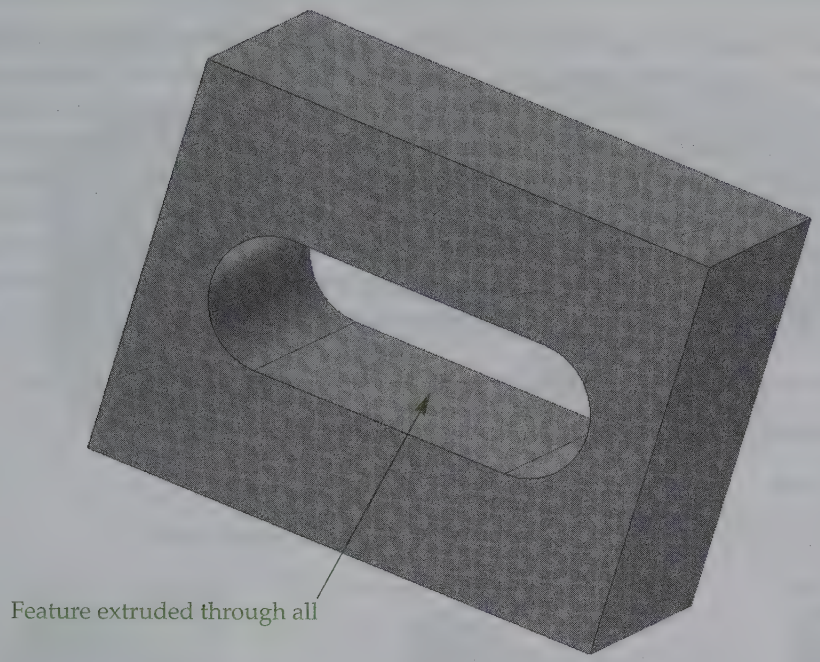
Figure 5-34.
The **From To** option of the **Extrude** dialog box, **Extents** area.



If you are having difficulty using the **From To** extent option, you may need to specify additional extent information in the **Alternate Solution** area of the **More** tab. Use the **Flip Direction** buttons to change the direction of the extrusion. You can also select the **Minimum Solution** check box to terminate the extrusion at the nearest plane or face instead of possibly cutting through multiple planes or faces.

- **All.** Selecting this option allows you to extrude a feature directly through all other features if you know for sure you want a feature to extrude all the way through existing model geometry. See Figure 5-35. **All** does not

Figure 5-35.
Creating an extrusion using the **All** termination option.



require a distance or any surface selections, and ensures that the extrusion is always applied through all features, even if you modify feature sizes later. To extrude a feature through all existing features, pick the sketch profile and select the direction you want the extrusion to occur. Then pick the **OK** button to create the extruded feature.

NOTE

Use any of the termination options previously described to end a feature at any feature, face, surface, or plane.

Exercise 5-6

Complete the Exercise on the Student CD.

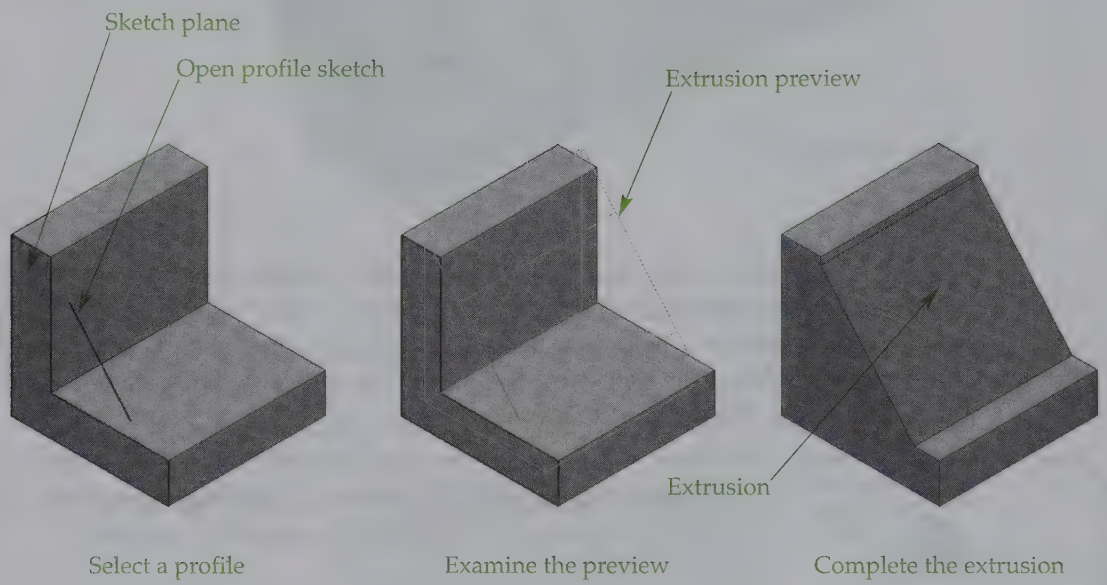
Creating Additional Revolutions and Coils

You have already learned how to develop revolutions and coils using the **Join** operation option. When additional features are present, you can either use the **Join**, **Cut**, or **Intersect** operations, depending on the design requirement. These operations function exactly the same for additional revolved and coil features as they do for additional extruded features.

Using Open Sketch Profiles

As you have learned, sketched features are typically created using closed profile sketches. However, in some situations it is possible to extrude or revolve an ***open sketch profile***, which is a sketch that does not form a closed loop. A sketch does not even have to touch or intersect other features in order to generate a sketch profile that can be used to form a feature, assuming enough reference geometry is available from other features. For example, Figure 5-36 shows a sketch of a line that is not dimensioned, is unconstrained, and does not create a closed loop. When you select the open sketch profile, a preview of a closed profile is automatically created based on the surrounding feature geometry. If this is acceptable, you can then generate the extrusion.

Figure 5-36.
These are the steps to create an extrusion from an open profile.



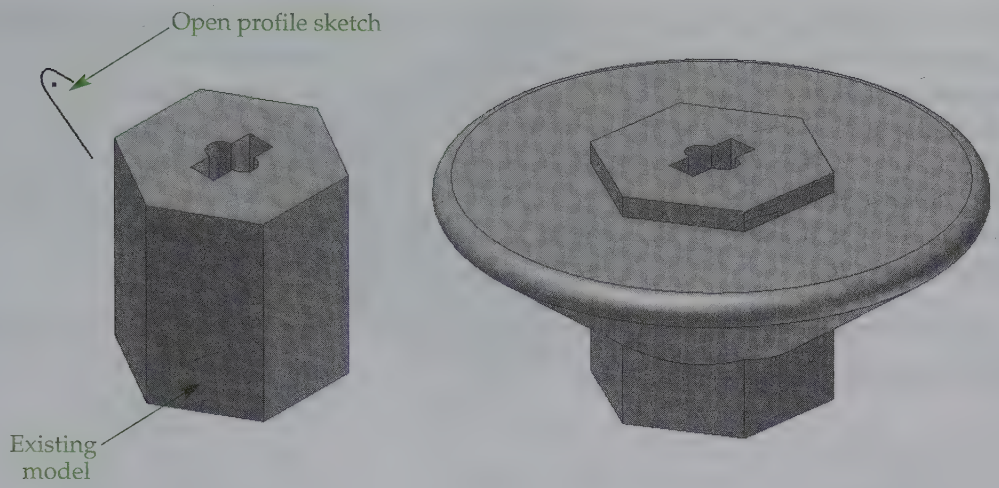
NOTE

Selecting an open profile is a two-step process. When you first select an open sketch profile, a preview of a closed profile is displayed. You must then pick the profile preview to make the selection.

Another example of using an open sketch profile is shown in Figure 5-37. In this example, a revolution is created using an open sketch of two lines and an arc. Notice how the revolution terminates itself in reference to the existing model.

You may have noticed the **Match shape** check box in the **Extrude** and **Revolve** dialog boxes, which is only available when working with open sketch profiles and can become very important when developing features. For example, the sketch shown in Figure 5-38 is an open profile that is intended to be used to form an extrusion that terminates at the surface of the existing feature. If the **Match shape** check box is not selected, the extrusion is defined by the edge. When the **Match shape** check box is selected, the extrusion is formed according to the shape of existing features.

Figure 5-37.
The open profile sketch extends to the existing model while creating a revolution.

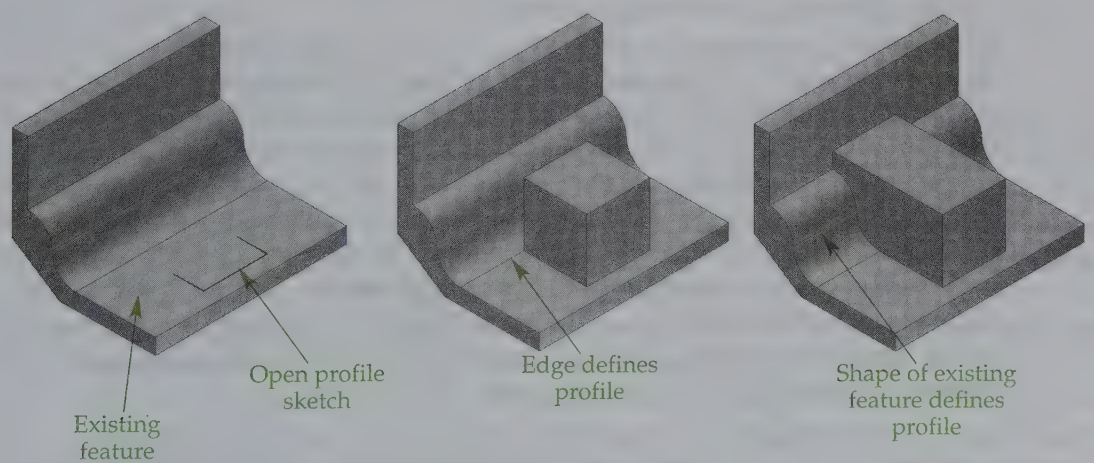


PROFESSIONAL TIP



An open sketch profile is best used to quickly develop additional extrusion sketches without having to project and reference existing edges in order to form a closed loop, and to terminate new features on existing shapes. Although it is possible to extrude and revolve an open, possibly unconstrained, sketch, you may still want to dimension and constrain your sketch in reference to existing features so that the sketch is defined and stable.

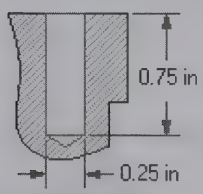
Figure 5-38.
This figure shows the difference between creating a feature using an open profile sketch with and without selecting the **Match shape** check box.



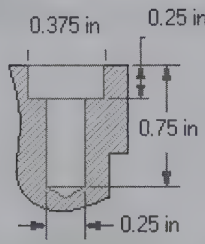
Creating Holes

Holes are used to create specific cylindrical extrusions through features. Applying holes is similar to extruding a circle, but the **Hole** tool provides you with many more options and allows you to create complex feature geometry in a single step. You can produce drilled, counterbored, and countersunk holes using the **Hole** tool. See Figure 5-39. ***Drilled*** holes are the simplest types of holes and require only one diameter and depth to create. ***Counterbored*** holes require a hole diameter and depth and a counterbore diameter and depth. ***Countersunk*** holes require a hole diameter and depth, and countersink diameter and angle.

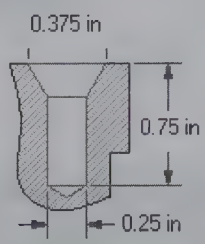
Figure 5-39.
Using the **Hole** tool to create various holes.



Drilled hole



Counterbored hole



Countersunk hole

NOTE

You can apply threads or specific fastener information to any of the three hole types previously described.

Access the **Hole** tool by picking the **Hole** button on the **Part Features** panel bar; picking the **Hole** button on the **Part Features** toolbar; or pressing the [H] key.

The location of holes can be defined using a sketch, existing feature edges, or a work point. You must establish how and where you want to place holes before they are created. The following information describes each of the hole placement options:

- **From Sketch.** This method requires that you make a sketch before you access the **Hole** tool. A sketch point created using the **Point, Hole Center** tool is often used to define the center of a hole. These points should be constrained and dimensioned in order to precisely locate the hole. You can also use any other point on a sketch, such as the end-point of a curve, or the center point of a fillet, arc, circle, or other radius. See Figure 5-40. When you place a hole using a sketch, the sketch becomes consumed by the hole, just as with other sketched features. When you sketch center points on a feature and access the **Hole** tool, the **From Sketch** option of the **Placement** drop-down list, shown in Figure 5-41, is used and sketch center points are automatically selected. If you want to choose other points, pick the **Centers** button and pick additional points.

NOTE

If you need to deselect points, pick the **Centers** button, hold down the [Ctrl] key, and choose the points.

Figure 5-40.
Sketching on a feature face in order to locate holes.

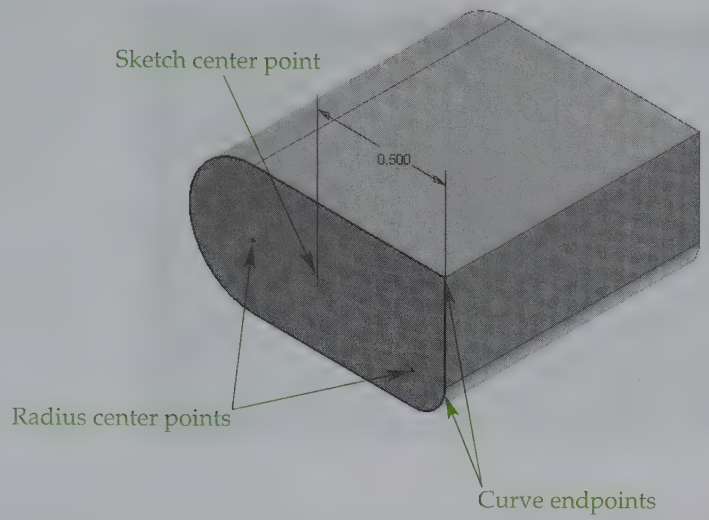
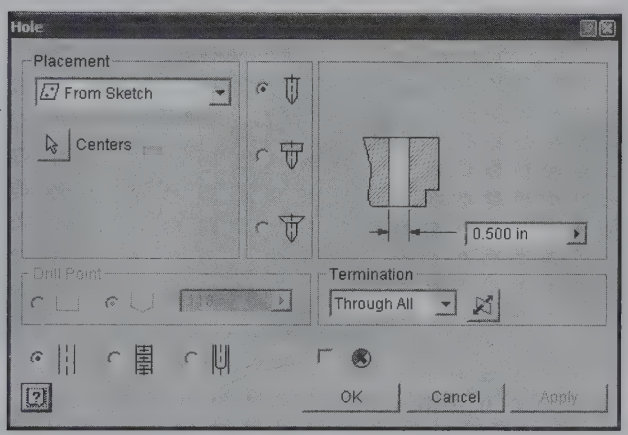


Figure 5-41.
The **Holes** dialog box, **From Sketch** placement option.



- **Linear.** This technique allows you to place a hole by referencing a feature face and two feature edges. The **Linear** method of placing holes is quick and easy for many applications because you can locate a hole exactly using linear dimensions, without using a separate sketch. When there is no sketch present and you access the **Hole** tool, the **Linear** option of the **Placement** drop-down list, shown in Figure 5-42, is used. The **Face** button should automatically be selected allowing you to choose the face on which the hole is placed. See Figure 5-43. Once you pick a feature face, the **Reference 1** select button becomes active. Use this button to select an edge on the existing feature to be used to define the location of the hole center. If the **Edit dimension when created** check box of the **Options** dialog box, **Sketch** tab is selected, the **Edit Dimension** dialog box appears, allowing you to define the distance from the edge to the hole center. If the **Edit dimension when created** check box is not selected, you have to double-click on the dimension after the second reference has been established, and modify the value. After you define the first reference, the **Reference 2** select button becomes active. Use this button to select an edge on the existing feature to be used to further locate the hole center. Again, if the **Edit**

Figure 5-46.
The **Holes** dialog box, **On Point** placement option.

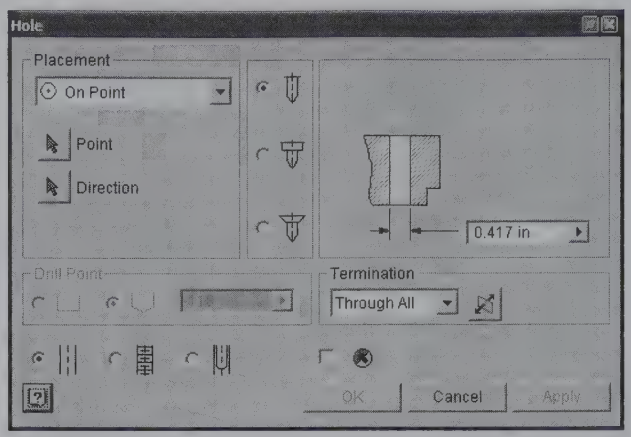
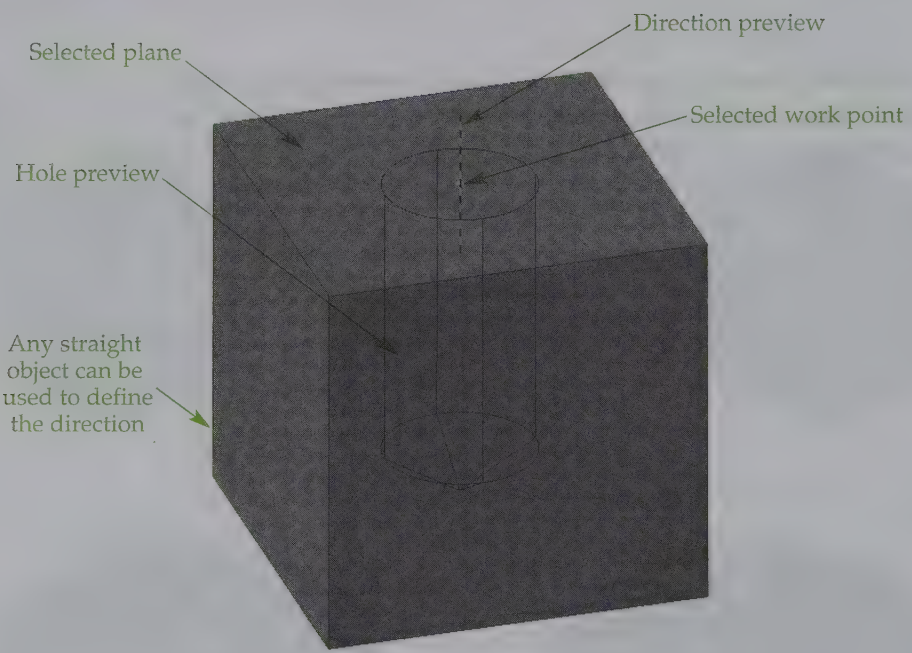


Figure 5-47.
On Point hole placement.

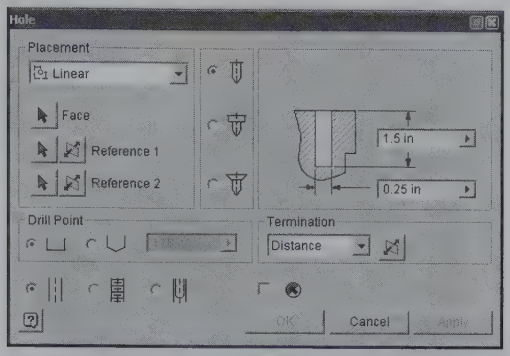


pick a work point, the **Direction** select button becomes active. Use this button to select an object to define the direction of the hole axis.

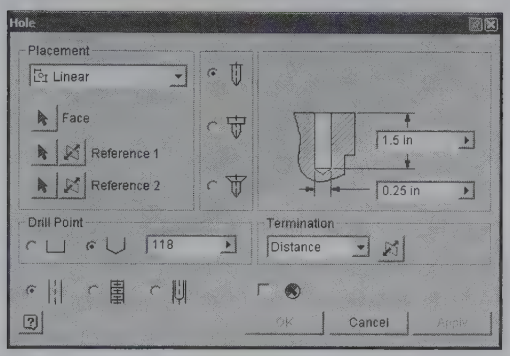
Once you locate the center of a hole, use the following options in the **Holes** dialog box to create hole features:

- **Drilled.** This radio button specifies the hole as drilled. Enter the hole diameter and depth using the edit boxes in the preview area.
- **Counterbore.** Pick this radio button to specify the hole as counterbored. Define the hole diameter and depth, and the counterbore diameter and depth inside the preview area using the appropriate edit boxes.
- **Countersink.** This button specifies the hole as countersunk. Enter the hole diameter and depth, the countersink diameter, and the countersink angle using the edit boxes in the preview area.
- **Drill Point.** Use this area to define the drill point characteristics for the selected holes. See Figure 5-48. Pick the **Flat** radio button to specify a flat 0 degree, drill

Figure 5-48.
Drill point options.



Flat



Angle

point. To specify an angled drill point, pick the **Angle** radio button and choose an angle from the list or specify the desired angle in the edit box.

- **Termination.** The termination area is very similar to the **Extents** area of the **Extrude** dialog box. As shown in Figure 5-49, the following termination options are available:
  - **Distance.** Use this choice to end a hole at a desired depth using the hole depth edit box in the preview area previously mentioned. If necessary, use the **Flip** button to flip the drill direction.
  - **Through All.** Selecting this option drills a hole directly through all features. If necessary, use the **Flip** button to flip the drill direction.
  - **To.** Allows you to choose a face or plane to end the drill. See Figure 5-50. To terminate a hole at a specified face or plane, pick the face or plane to

Figure 5-49.
Hole termination options include **Distance**, **Through All**, and **To**.

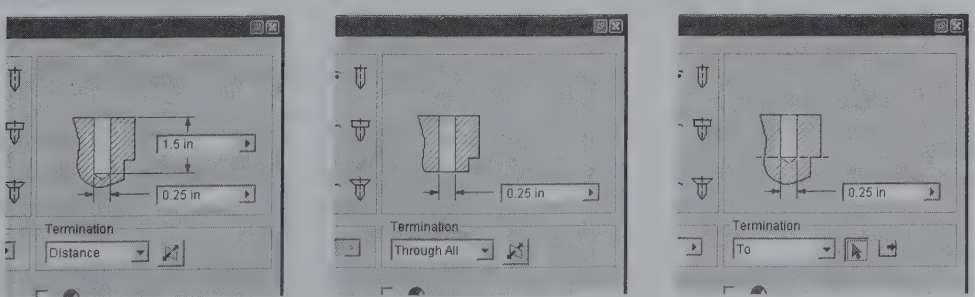
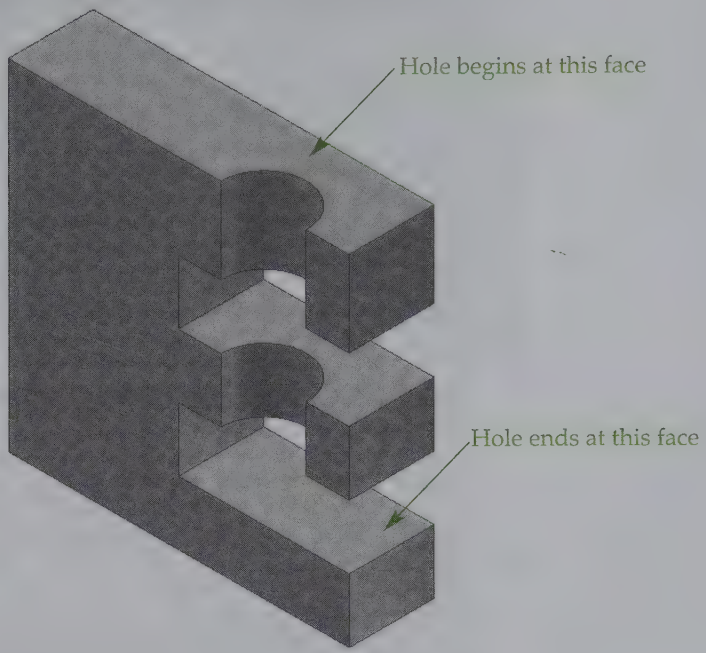


Figure 5-50.
Creating a hole using the **To** termination option (part is split for clarity).



which you want the hole to end. You may need to choose the **Select surface to end the feature creation** button if it is not already activated. If the face or plane you choose does not intersect the hole path, you need to check the **Terminate feature on extended face** check box.

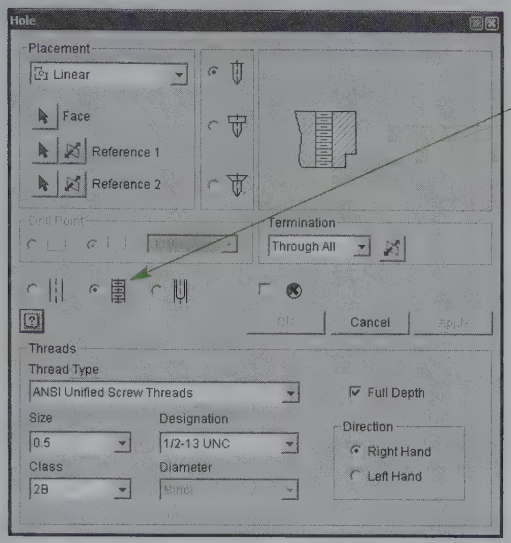
NOTE



Many, if not all, of the preview area edit boxes are eliminated when you create tapped or clearance holes, and holes with certain termination, because the sizes are defined by the thread, clearance, and termination settings.

- **Simple Hole.** Pick this radio button to create holes that have no threads and are not associated with a specific fastener.
- **Tapped Hole.** Pick this radio button to tap, or add threads to, the hole features. When you select the **Tapped Hole** radio button, the **Threads** area of the **Holes** dialog box is displayed. See Figure 5-51. The **Threads** area contains the following options that allow you to define the thread characteristics of a tapped hole:
  - **Full Depth.** Pick this check box to apply threads to the entire depth of the hole. By default the **Full Depth** check box is selected. If you do not want to tap the full length of the hole, deselect the **Full Depth** check box and specify the thread depth using the corresponding edit box in the preview area.
  - **Thread Type.** This drop-down list allows you to specify the thread standard used to create the tapped hole. You can choose ANSI Unified Screw Threads for inch parts, ANSI Metric M Profile for metric parts, or a variety of other thread standards, including ISO, DIN, BSP, and JIS.
  - **Direction.** Use this area to specify whether the threads are right or left hand by choosing the appropriate radio button. Right-hand threads move a right-hand

Figure 5-51.
The **Holes** dialog box, **Tapped Hole** radio button selected.



threaded bolt forward in a clockwise fashion, while left-hand threads move a left-hand threaded bolt forward in a counterclockwise fashion.

- **Size.** This drop-down list allows you to specify the nominal size of the tapped hole. The nominal size is the designated size of a commercial product and is usually the same as the major diameter.
- **Designation.** This drop-down list is used to specify the thread description, including pitch. ***Pitch*** is the distance parallel to the axis between a point on one thread to the corresponding point on the next thread. For ANSI Unified Screw Threads, the pitch determines and is specified by the number of threads per inch. For ANSI Metric M Profile threads, the pitch is specifically defined as the pitch.

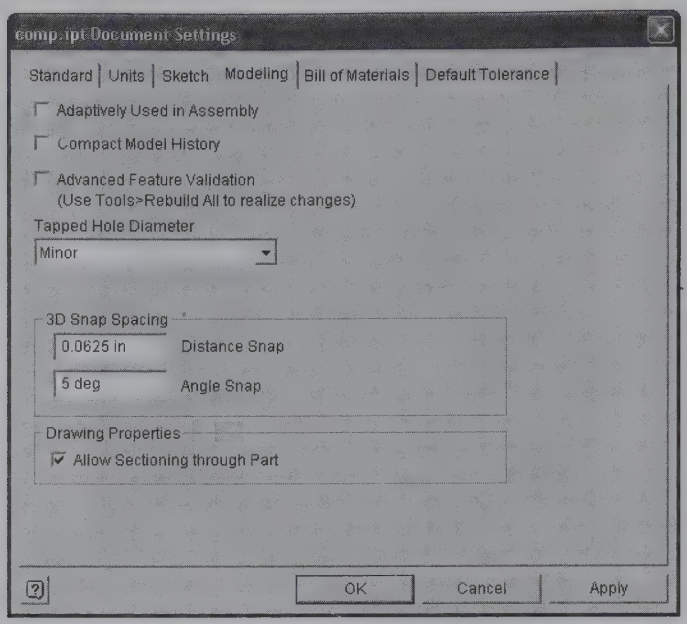
NOTE



Only designation options that correspond to the selected size are available.

- **Class.** This drop-down list allows you to specify the thread class. ***Class*** is the designated amount of tolerance specified for the thread. For ANSI Unified Screw Threads, 3, 4, 5, 6, 7, 8, or 9 can be available depending on the selected size and designation. Class identifies the grade of tolerance from fine to coarse threads. For ANSI Metric M Profile threads, 1, 2, or 3 can be available depending on the selected size and designation. 1 is a coarse tolerance, 2 is a moderate tolerance, and 3 is a fine thread tolerance. For holes, these numbers are followed by the letter B, because B identifies internal threads.
- **Diameter.** This drop-down list identifies the diameter of the nominal size as **Minor**, **Pitch**, **Major**, or **Tap Drill**. The **Diameter** drop-down list functions as a display box, because the value is actually specified using the **Tapped Hole Diameter** drop-down list in the **Modeling** tab of the **Document Settings** dialog box. See Figure 5-52. Though the nominal size of a hole is typically defined by the major diameter, you must select the **Minor** option from the **Tapped Hole Diameter** drop-down list in order for threads to be correctly represented in a drawing.

Figure 5-52.
The **Modeling** tab of the **Document Settings** dialog box.

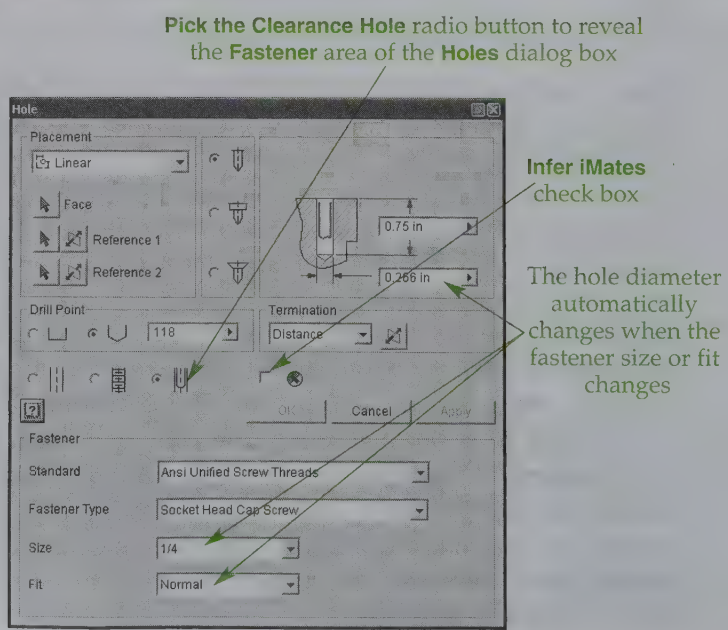


NOTE

When you apply threads to a hole, a bitmap thread image is placed on the hole walls. These thread representations are adequate for most applications, but if you need to create actual detailed threads, you must use the **Coil** tool.

- **Clearance Hole.** Pick this radio button to define the size and shape of a hole according to a specific type of fastener. When you select the **Clearance Hole** radio button, the **Fastener** area of the **Holes** dialog box is displayed. See Figure 5-53.

Figure 5-53.
The **Holes** dialog box, **Clearance Hole** radio button selected.



The **Fastener** area contains the following options:

- **Standard.** This drop-down list allows you to specify the thread standard used to create the hole. You can choose **ANSI Unified Screw Threads** for inch parts, **ANSI Metric M Profile** for metric parts, or a variety of other thread standards, including **ISO**, **DIN**, **BSP**, and **JIS**.
- **Fastener.** Use this drop-down list to pick a specific type of fastener based on the selected standard.
- **Size.** This drop-down list contains multiple fastener size options based on the fastener type to select.
- **Fit.** Use this drop-down list to select a **Close**, **Normal**, or **Loose** fit depending on the application.

NOTE

The **Infer iMates** check box is available when creating holes to add iMates to circular hole edges. iMates can be used to assemble components.

Exercise 5-7

Complete the Exercise on the Student CD.

Creating Ribs and Webs

Ribs and ***webs*** are thin sections of material placed between part features to reinforce the part without adding too much weight. Ribs and webs are similar and can be created from the same sketch geometry. Ribs are closed features, while webs are open. The ribs and webs displayed in Figure 5-54 constitute a ***rib*** and ***web network***. A rib or web network consists of several ribs and webs created using the same direction and thickness.

NOTE

A rib or web network can be created using a single sketch, as shown in Figure 5-54, or a patterned feature.

The first step in creating a rib or web is developing a sketch. Rib and web sketches can be open or closed looped, and can be fully or partially constrained, or totally unconstrained. See Figure 5-55. However, typically you may still want to situate your sketch in reference to existing features so your sketch is defined and stable. See Figure 5-56. If you are creating a rib or web network, sketch all profiles on the same sketch plane.

Once you develop a sketch, access the **Rib** tool by picking the **Rib** button on the **Part Features** panel bar, or picking the **Rib** button on the **Part Features** toolbar, to open the **Rib** dialog box. See Figure 5-57. To create a rib or web, pick the **Profile** button, if it is not active, and select the sketch profile. Then pick the **Direction** button and specify where you want the rib or web to occur. Direction arrows and a preview of the rib or web are shown when you move your cursor. See Figure 5-58. If the arrows are pointing to the correct location, and the preview looks okay, select the direction. Next, specify a rib or web width from the **Thickness** drop-down list, or enter the desired value. Then pick the side of the profile on which you want the rib to be created by selecting one of

Figure 5-54.
An example of ribs and webs created from the same sketch.

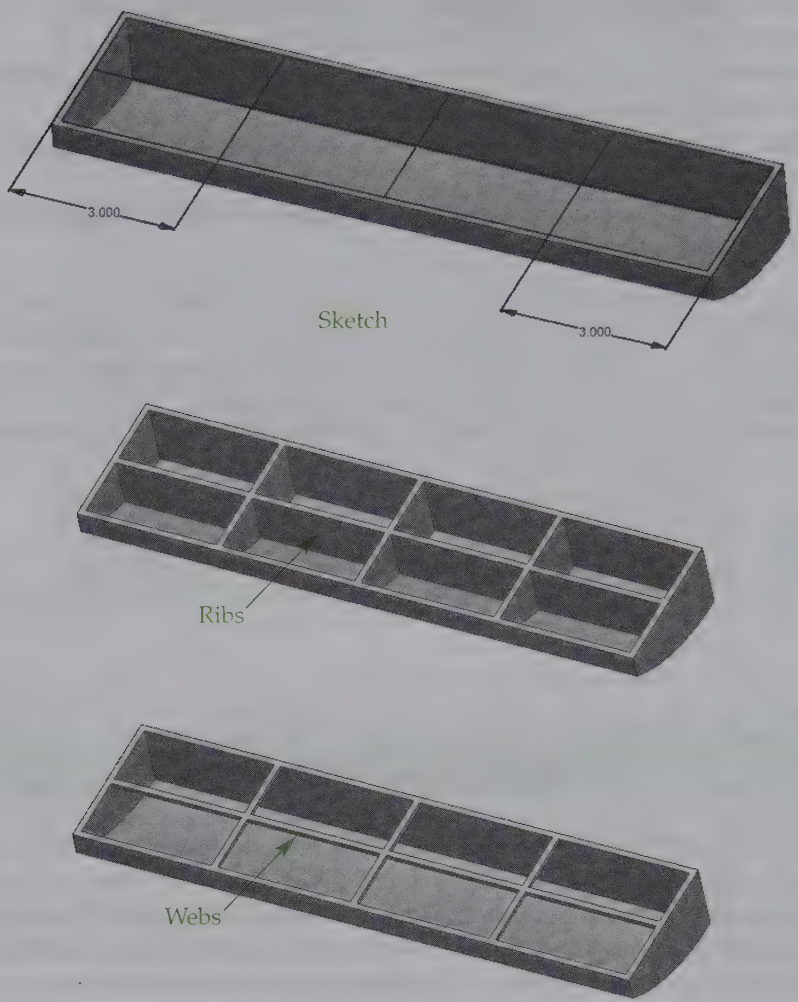


Figure 5-55.
An example of an open loop, unconstrained sketch used to create ribs and webs.

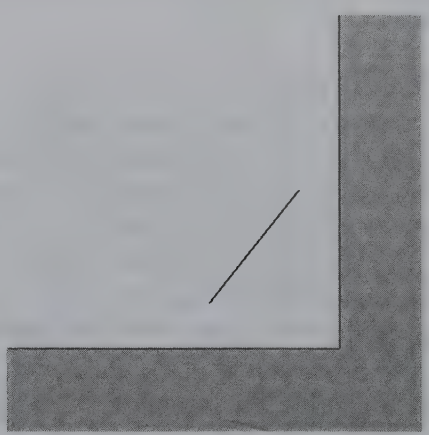


Figure 5-56.
An example of a constrained sketch, referencing existing geometry, used to create ribs and webs.

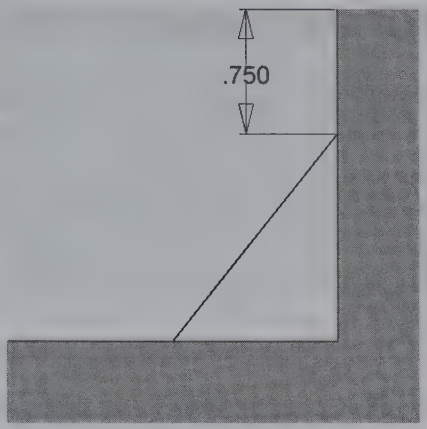


Figure 5-57.
The **Rib** dialog box, **To Next** extents button selected.

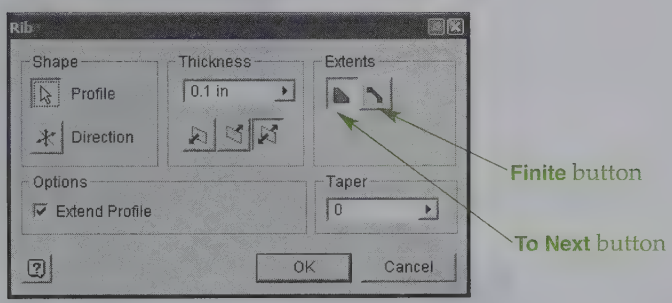


Figure 5-58.
Rib direction arrow and preview.

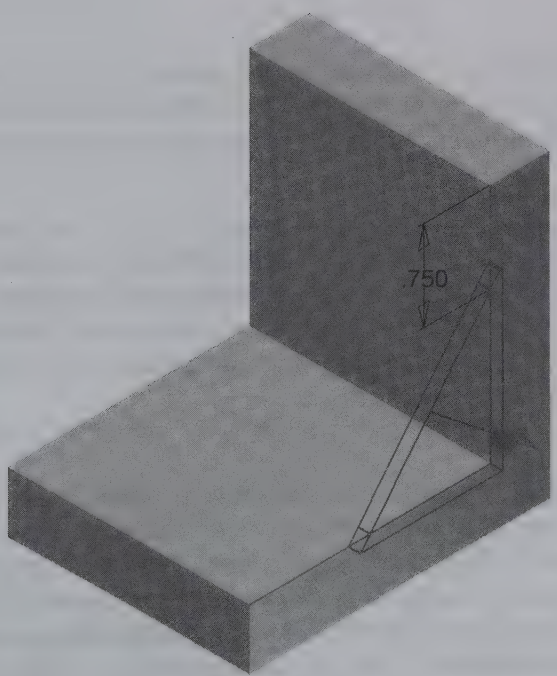


Figure 5-59.
The **Rib** dialog box, **Finite** extents button selected.

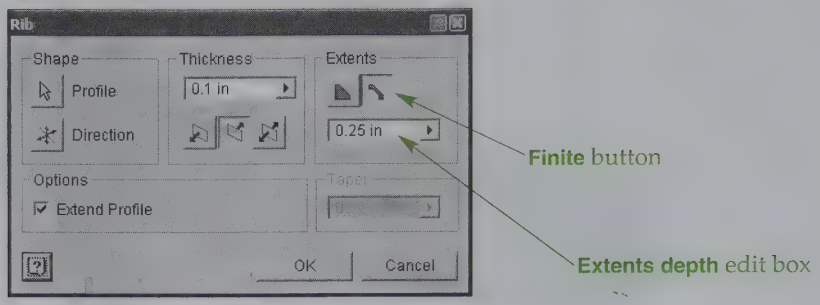
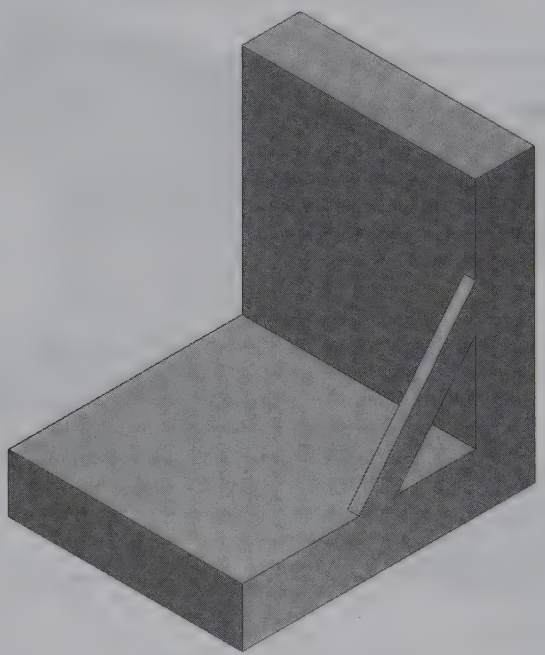


Figure 5-60.
An intersecting web created by extending the sketch profile.



the side buttons. You also have the option of tapering the rib by entering a taper angle in the **Taper** edit box. By default, the **To Next** extent button is active when you open the **Rib** dialog box. As a result, when all the preferences are set, pick the **OK** button to create a rib.

NOTE



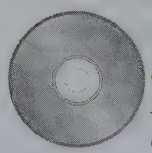
Occasionally, it is difficult to locate the direction you want the rib or web to be created. Do not attempt to generate a rib or web until a full preview of the feature is displayed and you are fairly sure the rib will be created successfully.

If you want to create a web, use the same process as previously described for creating a rib. However, you must pick the **Finite** extents button. When the **Finite** button is active, as shown in Figure 5-59, an **Extents Depth** edit box and an **Extend Profile** check box are displayed. The **Extents Depth** edit box allows you to specify the web thickness, or depth. Define the depth by choosing or entering a value in the edit box. If selected, the **Extend Profile** check box allows you to extend the ends of the web to the

intersecting features. See Figure 5-60. If unselected, the web terminates at the ends of the sketch profile. After you specify all the web options, pick the **OK** button to create the web.



Exercise 5-8 Complete the Exercise on the Student CD.



Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. What are sketched features created from, and generally, how complex are sketched features?
2. Identify the name of the initial feature that is often created from a sketch.
3. When can you create sketched features?
4. Name at least four sketched features.
5. Name one of the most basic and often least complicated Inventor features.
6. Compare a *revolved* sketched feature to an *extrusion*.
7. How are coils different from revolved features, and give an example of parts made from coils?
8. Define *pitch*.
9. What is a *revolution* when referring to coils?
10. How do you use the **Autoproject edges during curve creation** tool, and what happens to the edge color when using this tool?
11. Name the simplest hole type that requires only one diameter and depth to create.
12. Name the hole type that requires a hole diameter and depth and an additional diameter and depth to create.
13. Name the hole type that requires a hole diameter and depth and an additional diameter and angle to create.
14. Define *ribs* and *webs* and explain the difference between each.
15. What is a *rib* or *web network*?

Problems

Instructions:

- Open the following Chapter 4 part files, or create new parts as needed, and follow the specific instructions provided for each problem.

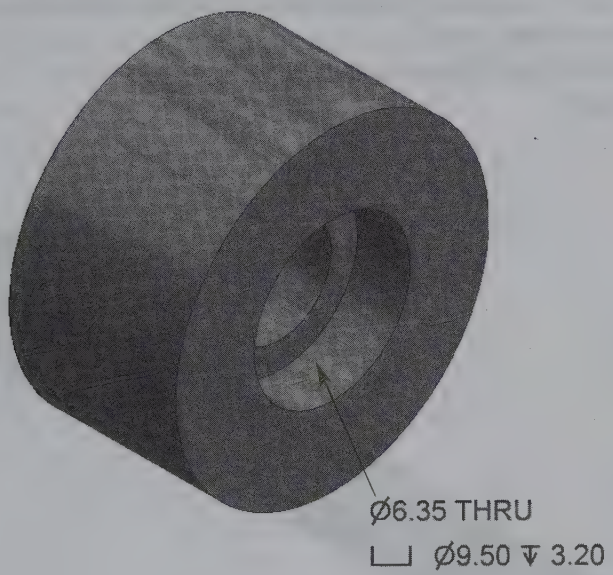
1. **Name:** C-Clamp Pin
   Units: Metric
   Template: PART-mm.ipt
   Material: Steel, High Strength Low Alloy
   Color: As Material
   Specific Instructions:
   - Open P4-1.ipt and save a copy as P5-1.ipt.
   - Close P4-1.ipt without saving and open P5-1.ipt.
   - Extrude the sketch profile 52mm in a positive direction as shown.

   Save As: P5-1.ipt



2. **Name:** C-Clamp Swivel
Units: Metric
Template: PART-mm.ipt
Material: Steel, High Strength Low Alloy
Color: As Material
Specific Instructions:
- Open P4-2.ipt and save a copy as P5-2.ipt.
- Close P4-2.ipt without saving and open P5-2.ipt.
- Extrude the sketch profile 9.5mm in a negative direction as shown.
- Use the **Concentric** placement option of the **Hole** tool to apply the counterbored hole shown, using the dimensions provided.

Save As: P5-2.ipt



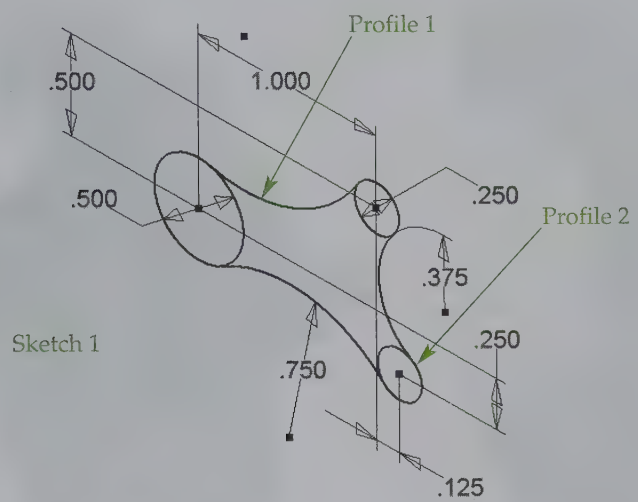
3. **Name:** C-Clamp Screw
Units: Metric
Template: PART-mm.ipt
Material: Steel, High Strength Low Alloy
Color: As Material
Specific Instructions:
- Open P4-3.ipt and save a copy as P5-3.ipt.
- Close P4-3.ipt without saving and open P5-3.ipt.
- Extrude the sketch profile 80mm in a positive direction as shown.
- Open a sketch on one end and sketch a ∅6.35 circle at the center point.
- Extrude the sketch 7.2mm as shown. Be sure you select both available sketch profiles.
- Open a sketch on the opposite end of the first extrusion, and sketch a ∅12.7 circle at the center point.
- Extrude the sketch 22mm as shown.

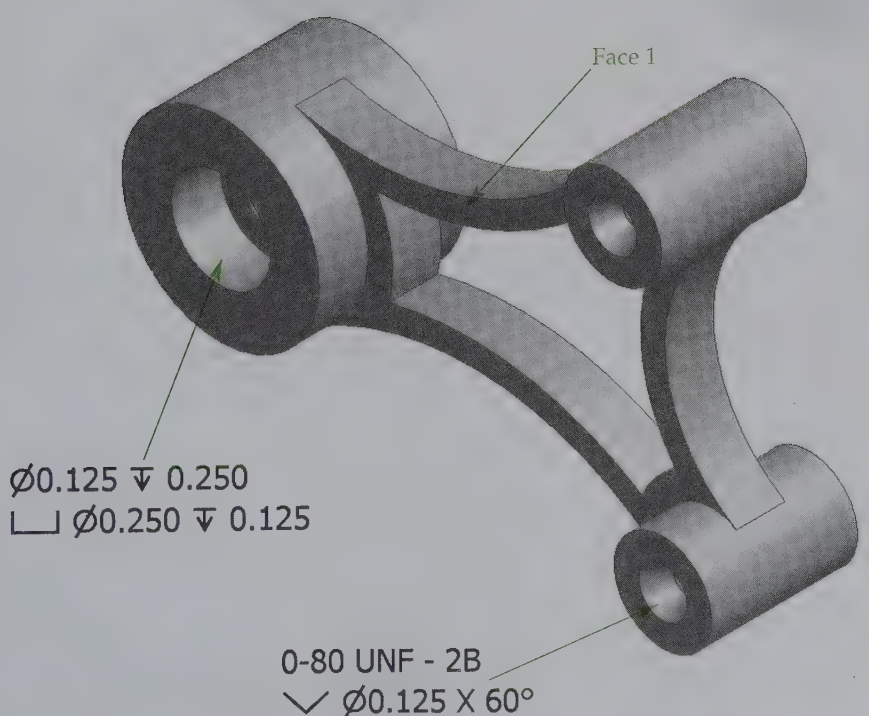
Save As: P5-3.ipt



4. **Name:** Support Bracket
Units: Inch
Template: PART-in.ipt
Material: Aluminum-6061
Color: As Material
Specific Instructions:
- Open P4-4.ipt and save a copy as P5-4.ipt.
- Close P4-4.ipt without saving and open P5-4.ipt.
- Extrude Profile1, .125" midplane.
- Share Sketch1.
- Extrude Profile 2, .375" midplane.
- Open a sketch on Face1, and offset the automatically projected geometry .0625". You may need to edit and constrain sketch geometry.
- Cut-extrude the sketch through the part as shown.
- Apply the holes shown using the dimensions provided.

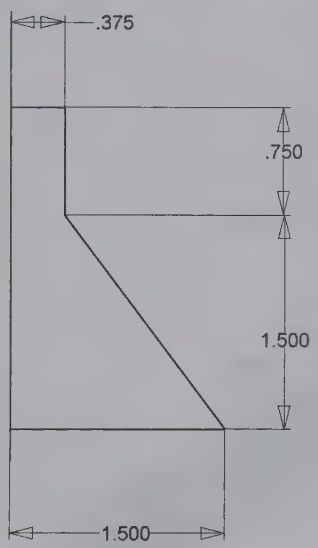
Save As: P5-4.ipt

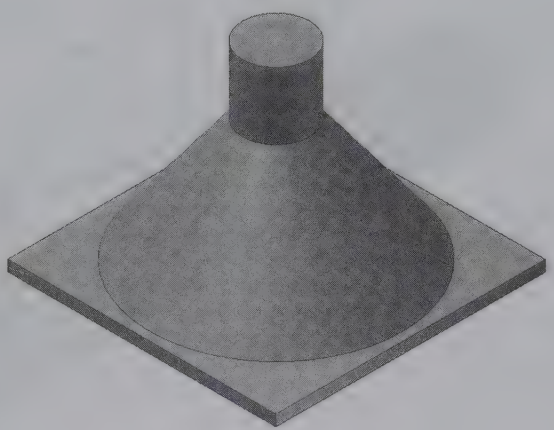




5. **Name:** Funnel
Units: Inch
Template: PART-in.ipt
Material: Acetal Resin, Black
Color: As Material
Specific Instructions:
- Begin a new inch part file.
- Create the sketch shown in the XY plane.
- Fully revolve the sketch around the Y axis.
- Sketch a rectangle tangent to the radius of the circle, on the base of the revolved feature.
- Extrude the sketch .125" in a negative direction, toward the revolution, as shown. Be sure you select both sketch profiles available.

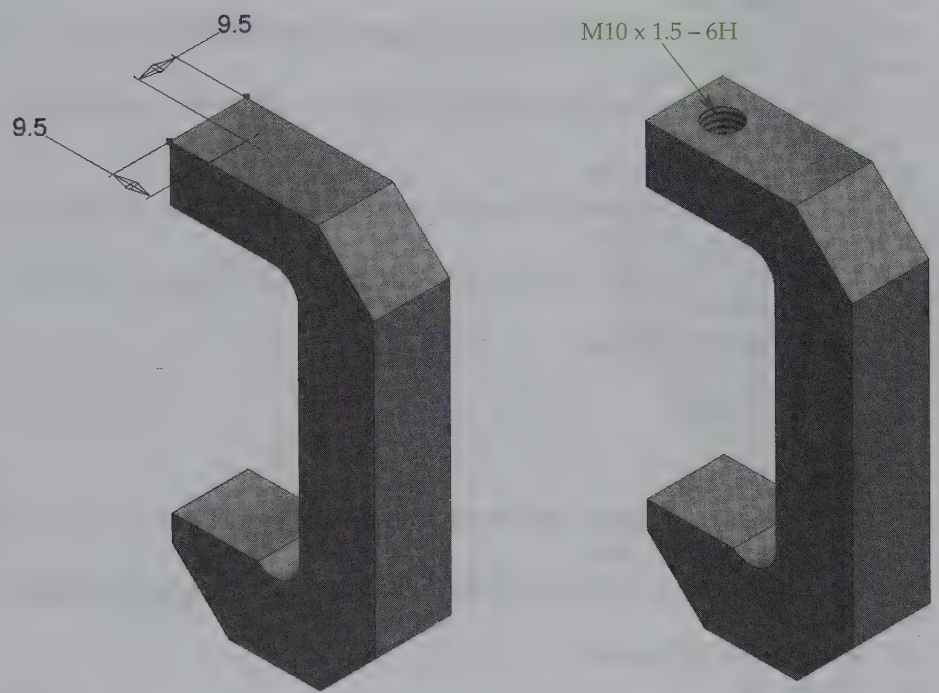
Save As: P5-5.ipt





6. **Name:** C-Clamp Body
Units: Metric
Template: PART-mm.ipt
Material: Steel, High Strength Low Alloy
Color: Red
Specific Instructions:
- Open P4-6.ipt and save a copy as P5-6.ipt.
- Close P4-6.ipt without saving and open P5-6.ipt.
- Extrude the sketch 19mm, midplane.
- Use the **Linear** placement option of the **Hole** tool to apply the hole shown using the dimensions provided.

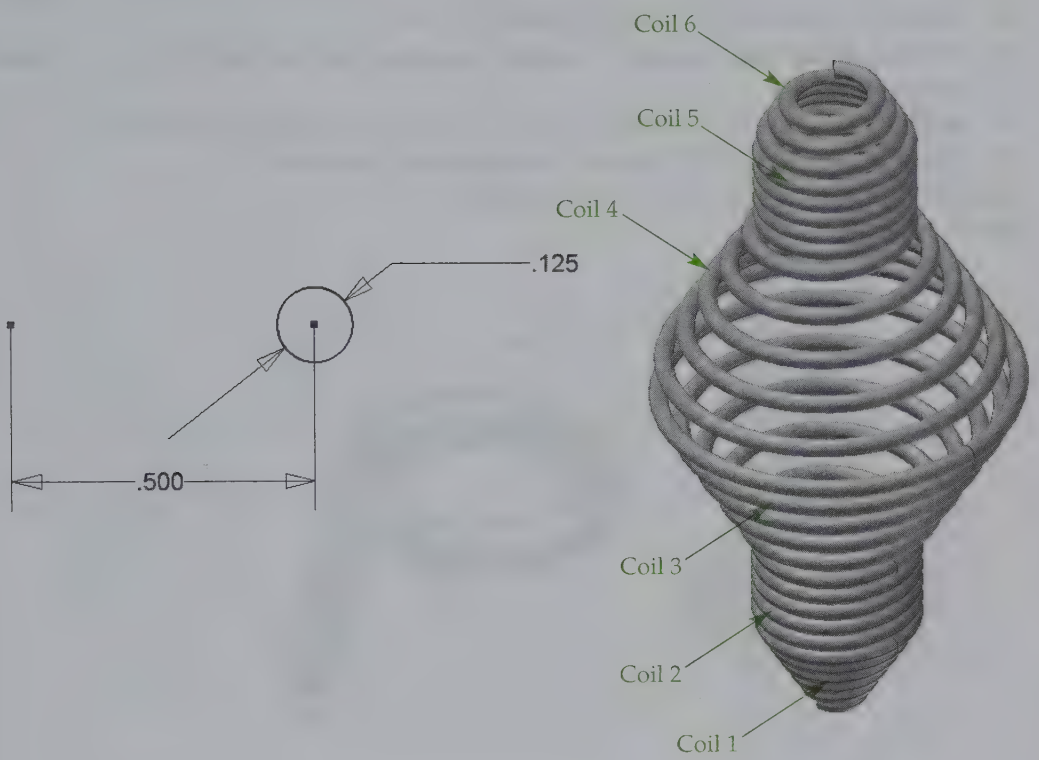
Save As: P5-6.ipt



7. **Name:** Torsion Spring
Units: Inch
Template: PART-in.ipt
Material: Default
Color: Default
Specific Instructions:
   - Begin a new inch part file.
   - Create the sketch shown on the XY plane.
   - Coil the sketch in a positive direction, using the following parameters:
     - **Axis:** Y axis
     - **Rotation:** Right-hand
     - **Pitch:** .125″
     - **Revolutions:** 5
     - **Ends:** Natural
   - Use the **Autoproject edges for sketch creation and edit** tool to create a sketch of the bottom coil face.
   - Coil the sketch in a negative direction, using the following parameters:
     - **Axis:** Y axis
     - **Rotation:** Right-hand
   - Use the **Autoproject edges for sketch creation and edit** tool to create a sketch of the top coil face of Coil 1.
   - Coil the sketch in a positive direction, using the following parameters:
     - **Axis:** Y axis
     - **Rotation:** Right-hand
     - **Pitch:** .25″
     - **Revolutions:** 5
     - **Taper:** 30°
     - **Ends:** Natural
   - Use the **Autoproject edges for sketch creation and edit** tool to create a sketch of the top coil face of Coil 3.
   - Coil the sketch in a positive direction, using the following parameters:
     - **Axis:** Y axis
     - **Rotation:** Right-hand
     - **Pitch:** .25″
     - **Revolutions:** 5
     - **Taper:** –30°
     - **Ends:** Natural
   - Use the **Autoproject edges for sketch creation and edit** tool to create a sketch of the top coil face of Coil 4.
   - Coil the sketch in a positive direction, using the following parameters:
     - **Axis:** Y axis
     - **Rotation:** Right-hand
     - **Pitch:** .125″
     - **Taper:** 0°
     - **Revolutions:** 5
     - **Ends:** Natural
   - Use the **Autoproject edges for sketch creation and edit** tool to create a sketch of the top coil face of Coil 5.

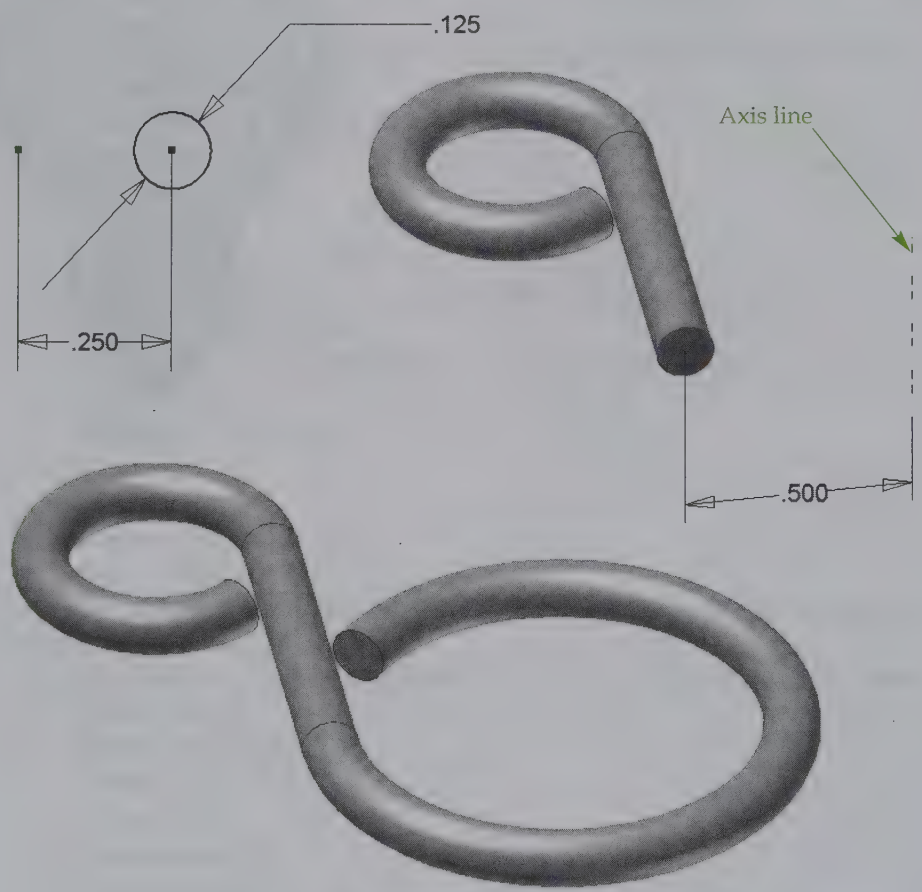
- Coil the sketch in a positive direction, using the following parameters:
  - **Axis:** Y axis
  - **Rotation:** Right-hand
  - **Pitch:** .125″
  - **Revolutions:** 3
  - **Taper:** –30°
  - **Ends:** Start=Natural, End=Flat with 45° flat angle.

The final project should look like the part shown.
Save As: P5-7.ipt



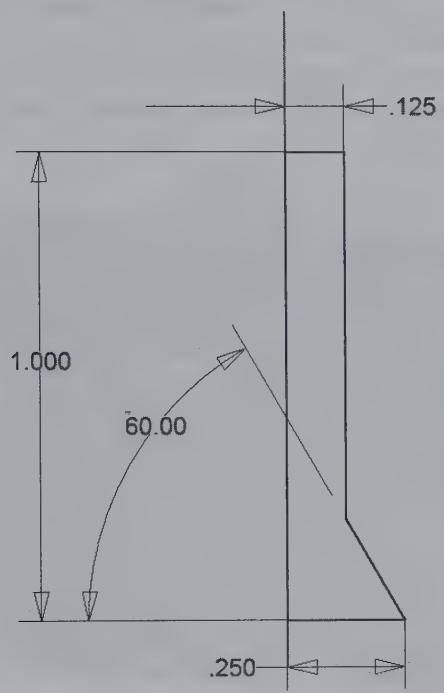
8. **Name:** Torsion Spring Swivel Eye
Units: Inch
Template: PART-in.ipt
Material: Default
Color: Default
Specific Instructions:
- Begin a new inch part file.
- Create the sketch shown on the default XY plane.
- Revolve the sketch 305° around the Y axis.
- Use the **Autoproject edges for sketch creation and edit** tool to create a sketch of Face1.
- Extrude the sketch .75″ in a positive direction.
- Use the **Autoproject edges for sketch creation and edit** tool to create a sketch of Face2.
- Add an axis line .5″ from the center of the sketched circle as shown.
- Revolve the sketch 320° around the sketched axis line.

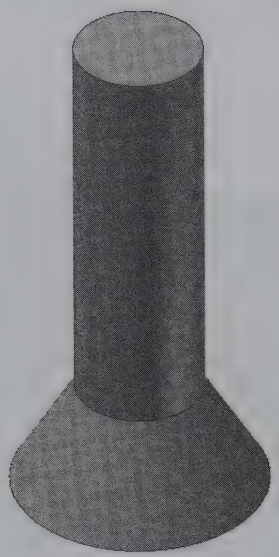
The final project should look like the part shown.
Save As: P5-8.ipt



9. **Name:** Torsion Spring Swivel Bolt
Units: Inch
Template: PART-in.ipt
Material: Default
Color: Default
Specific Instructions:
- Begin a new inch part file.
- Create the sketch shown on the XY plane.
- Fully revolve the sketch around the Y axis.

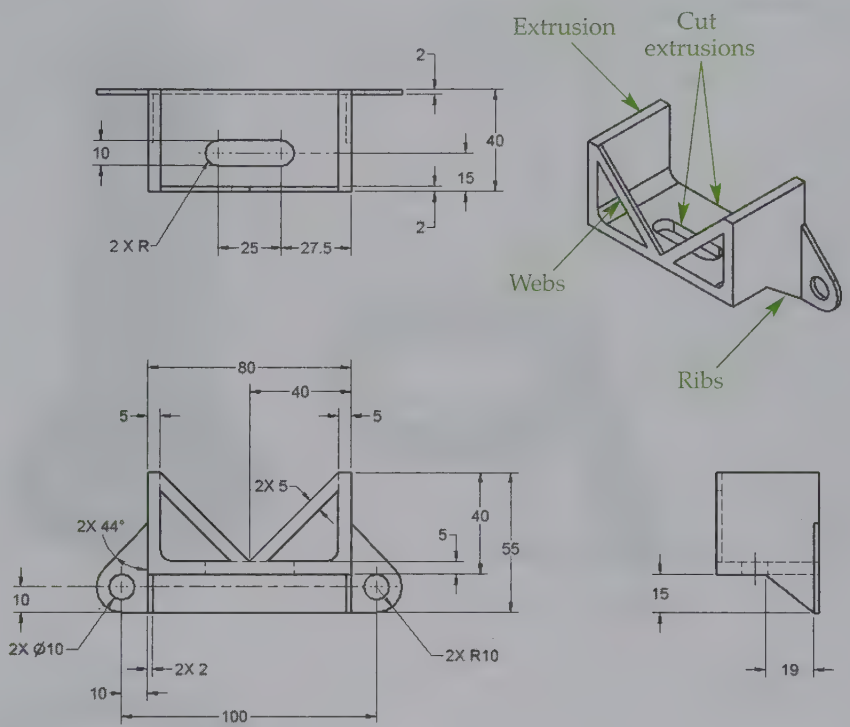
The final project should look like the part shown.
Save As: P5-9.ipt





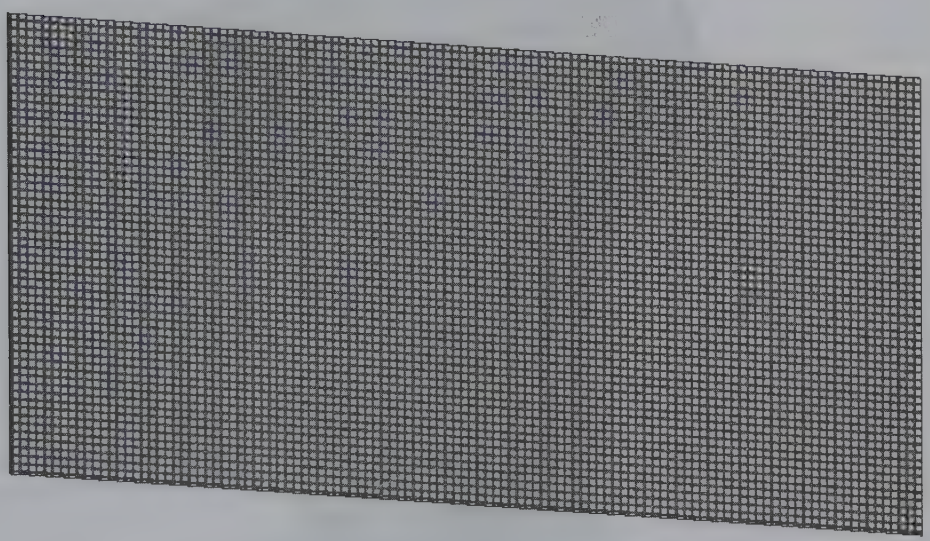
10. **Name:** Bracket
Units: Metric
Template: PART-mm.ipt
Material: Aluminum-6061
Color: Blue Sea
Specific Instructions:
- Begin a new part file.
- Create the part shown using the features specified.
- Open the **Properties** dialog box for the model and enter BRACKET in the **Title** edit box, and P5-10 in the **Part Number** edit box.

When complete, your part should contain at most 3 extrusions from 3 sketches, 3 ribs from 3 sketches, 2 webs from 1 sketch, and 2 holes from 1 sketch.
Save As: P5-10.ipt



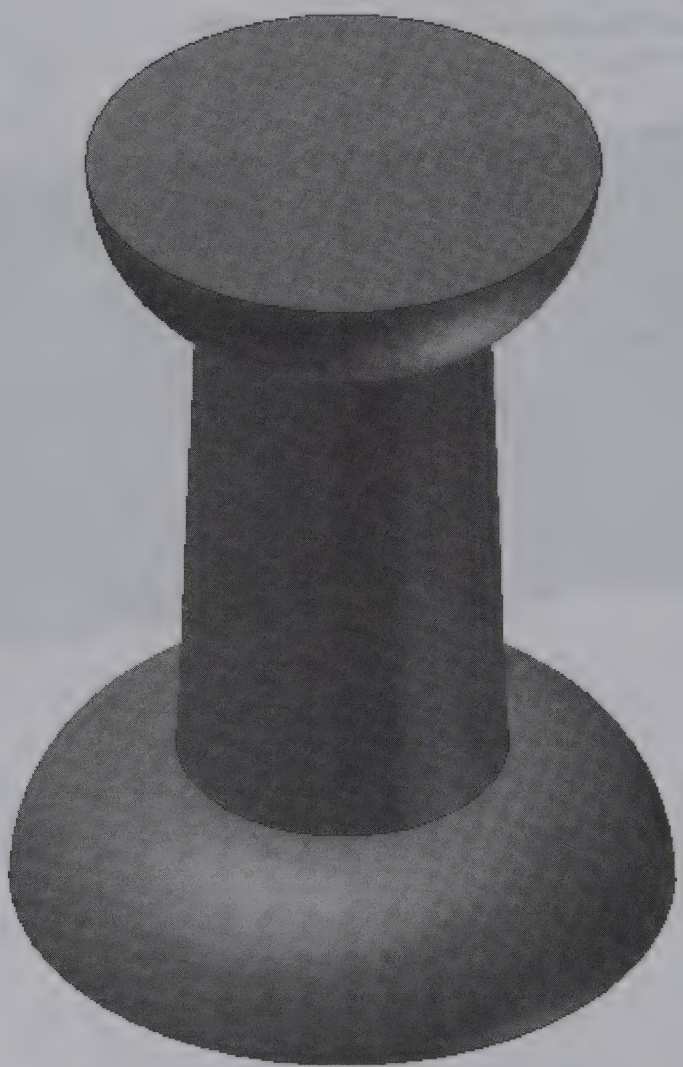
11. **Name:** Screen
Units: Inch
Template: PART-in.ipt
Material: Default
Color: Screen (created in the problem)
Specific Instructions:
- Begin a new inch part file.
- Extrude a 24″ × 48″ rectangle, .0375″ in a positive direction.
- Access the **Styles and Standards Editor**.
- Select Metal-Steel from the **Color** node.
- Pick the **New...** button and change the name of Copy of Metal-Steel to Screen.
- Pick the **Choose** button and select the Screen_3& texture style.
- Change the **% Scale** to 400%.
- Save and close the **Styles and Standards Editor**.
- Change the color of the part to Screen.

The final part should look like the screen shown.
Save As: P5-11.ipt



12. **Name:** Push Pin Head
Units: Inch
Template: PART-in.ipt
Material: ABS Plastic
Color: Green
Specific Instructions:
- Open P4-8.ipt and save a copy as P5-12.ipt.
- Close P4-8.ipt without saving and open P5-12.ipt.
- Revolve the sketch 360° around the Y axis.

The final part should look like the screen shown.
Save As: P5-12.ipt





CHAPTER 6

Placed Features

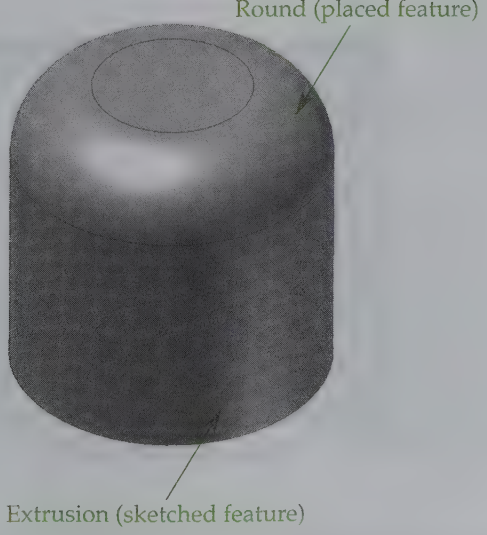
Learning Objectives

After completing this chapter, you will be able to do the following:

- Add fillets and rounds.
- Add chamfers.
- Shell features.
- Apply face drafts.
- Place threads on cylindrical and conical features.
- Thicken and offset faces or surfaces.
- Move faces.

Once you create a sketch and generate one or more sketched features, you are ready to add placed features. A ***Placed feature*** is created without a sketch and is added to an existing sketched, placed, catalog, or pattern feature. Placed features include fillets, chamfers, shells, face drafts, threads, thickened or offset features, and face moves. Placed features can be added to the most basic feature, such as an extruded rectangle, or to a very complex, multi-feature part. The **Fillet** tool was used to place the round on the extruded cylinder edge shown in Figure 6-1.

Figure 6-1.
An example of a round created using the **Fillet** placed feature tool.



Adding Fillets and Rounds

A *fillet* is a curve placed at the inside intersection of two or more faces, adding material to a feature. Typically, fillets are added to help release patterns from forgings and castings. They are also created to relieve stress and ease the machining of inside corners. A *round* is a curve placed on the exterior intersection of two or more faces, removing material from a feature. See Figure 6-2. Rounds remove sharp machined edges and help release patterns from forgings and castings. Both fillets and rounds are created using the **Fillet** tool, which is accessed by picking the **Fillet** button on the **Part Features** panel bar; picking the **Fillet** button on the **Part Features** toolbar; or pressing [Shift]+[F].

The **Fillet** dialog box is displayed when you access the **Fillet** tool. See Figure 6-3. Fillets and rounds can be placed on edges or can be formed according to selected faces. The first step to adding fillets or rounds is to choose the appropriate fillet and round type button in the **Fillet** dialog box. The following information describes each fillet and round type.

Figure 6-2.
The fillets and rounds in these parts were added with the **Fillet** tool.

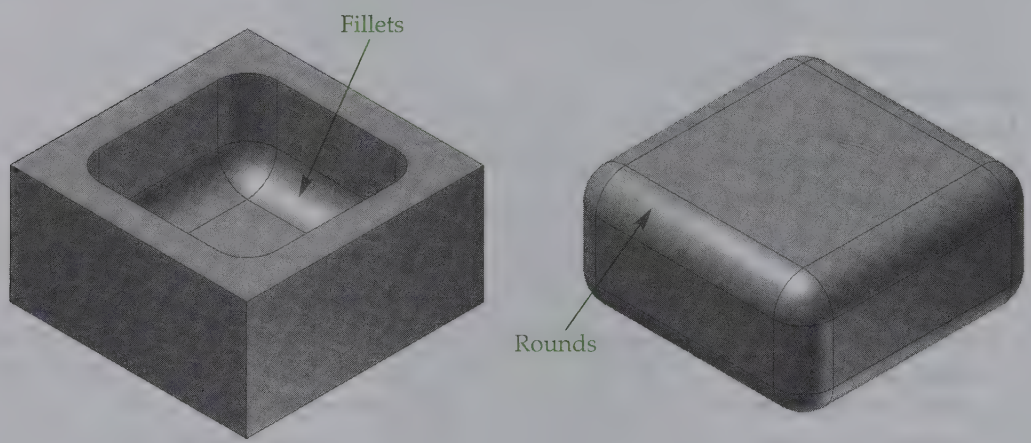
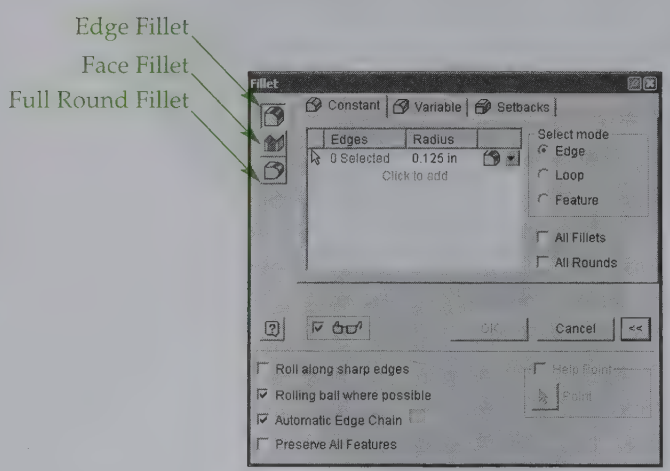


Figure 6-3.
The **Fillet** dialog box, **Constant** tab.



Creating Edge Fillets and Rounds

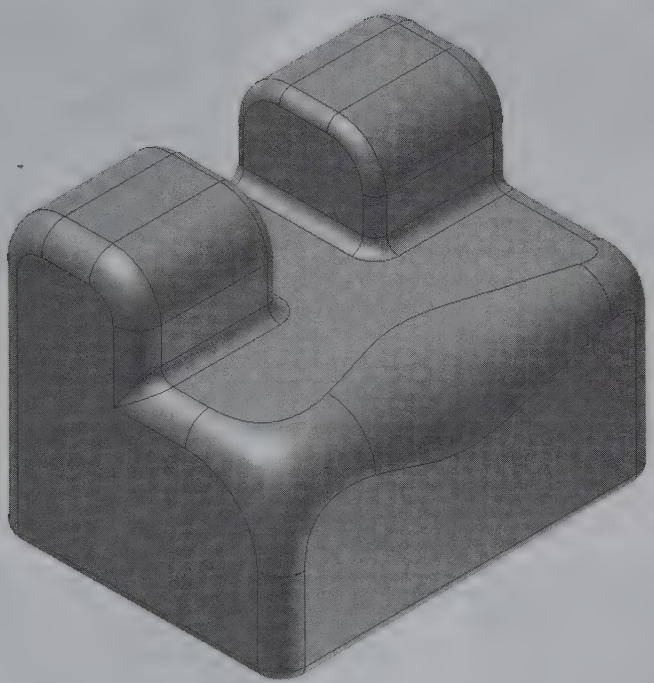
Pick the **Edge fillets** button in the **Fillet** dialog box to create fillets and rounds on selected feature edges. The **Edge Fillet** option of the **Fillet** dialog box can often be used to completely fillet and round a part using different fillet styles and radii. You do not need to continually access the **Fillet** tool to create a different type of fillet or round. Figure 6-4 shows an example of a part with different fillet and round styles and radii. All the fillets and rounds were created using a single fillet tool, and each of the fillets placed during one operation creates a single feature in the browser.

PROFESSIONAL TIP



If you plan to suppress only certain fillets or want to have additional filleted features, you may want to access the **Fillet** tool more than once.

Figure 6-4.
By accessing the **Fillet** tool once, you can apply several fillet and round styles, all having different radii.



Before you begin to create a fillet or round, you may want to adjust some of the options located in the **More** area of the **Fillet** dialog box. The following check boxes are available when you pick the **More** button:

- **Roll along sharp edges.** When this check box is not selected, a lip is formed when filleting an object around a sharp edge, which changes the fillet radius. If you want the fillet radius to remain constant and feature faces to stay the same, select the **Roll along sharp edges** check box. See Figure 6-5.
- **Rolling ball where possible.** This check box allows you to specify the corner style of fillets and rounds. See Figure 6-6. When the **Rolling ball where possible** check box is selected, fillets and rounds are placed on feature edges in a way that looks like a ball rolled around the edge. If the **Rolling ball where possible**

Figure 6-5.
Notice the change in the edge when the **Roll along sharp edges** check box is not activated.

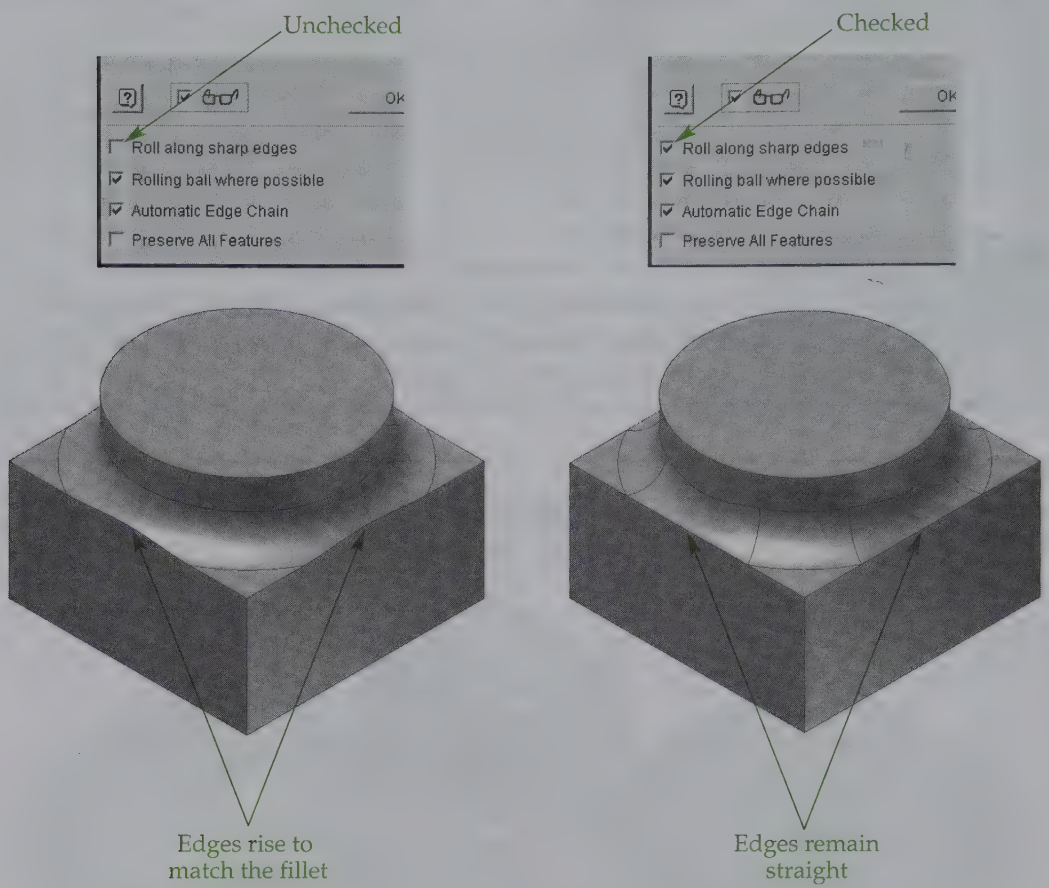
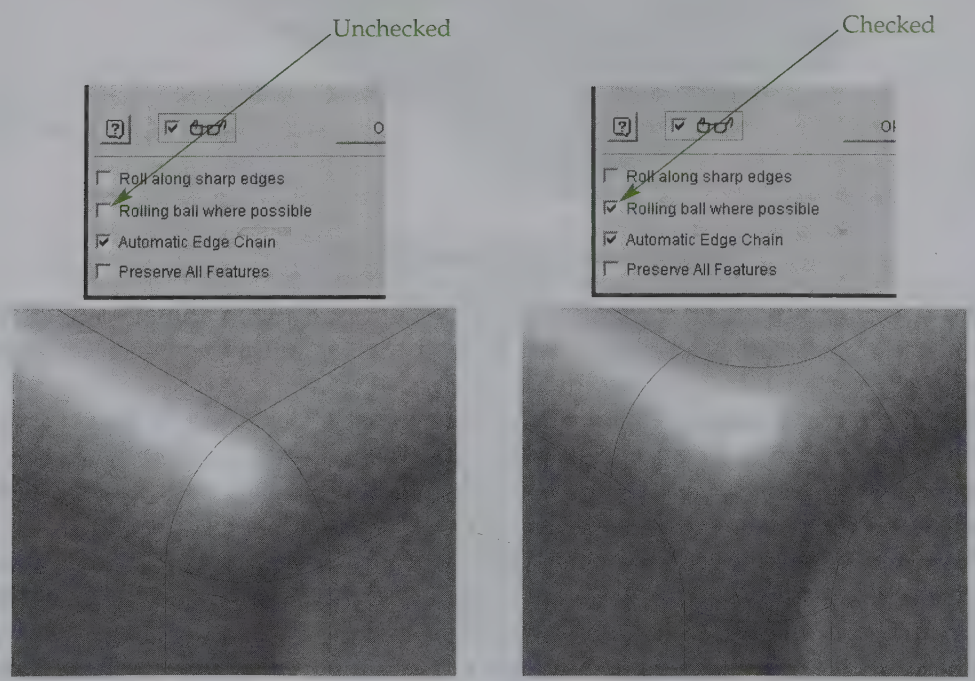


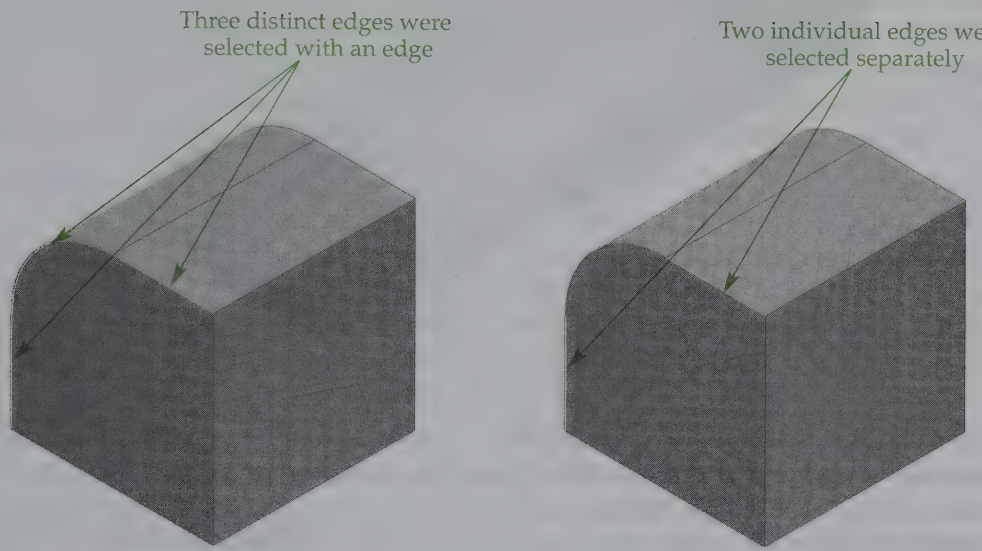
Figure 6-6.
Compare the two corners when the **Rolling ball where possible** check box is and is not activated.



check box is unselected, fillets and rounds are blended together, assuming at least one X, Y, and Z axis edge is selected.

- **Automatic Edge Chain.** This option allows you to select the entire edge chain of a tangent loop. When the **Automatic Edge Chain** check box is not selected, you can pick individual tangent loop edge segments. See Figure 6-7.
- **Preserve all features.** When this check box is selected, Inventor ensures that the features that intersect the fillet are accounted for, and that the intersection redefinition information is determined when the fillet is created.

Figure 6-7.
You can select with an edge chain to pick all tangent edges or select individual edges.



NOTE

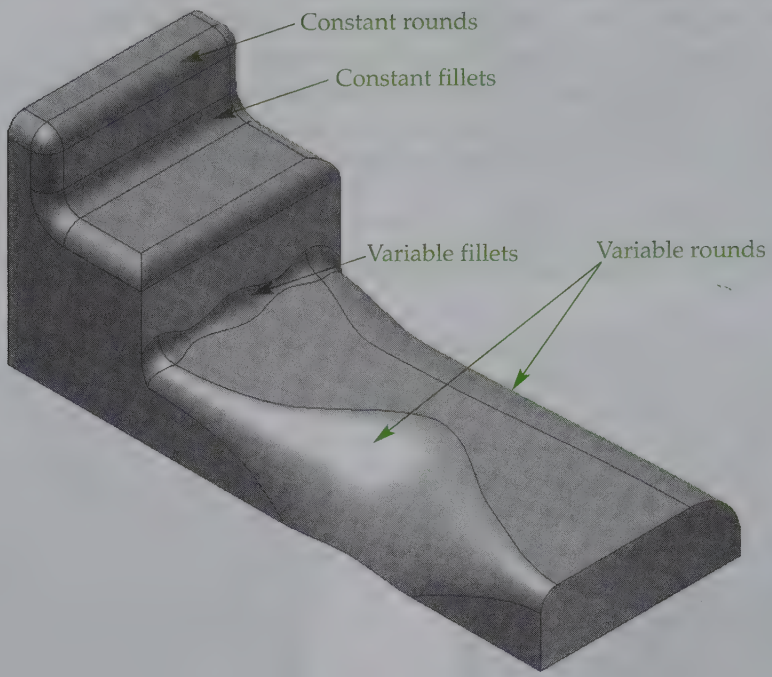
Typical fillets and rounds are created using the default selected check boxes in the **More** area of the **Fillet** dialog box. Still, the additional options may be required for some applications.

Edges fillets and rounds can be created as constant and variable. See Figure 6-8. A ***constant fillet*** or ***round*** has a curve radius that does not change. Constant fillets and rounds are the simplest fillet and round features to place. A ***variable fillet*** or ***round*** has different sized curved radii placed at precise points between the start and end of a feature edge.

To apply a constant fillet or round, use the options available from the **Constant** tab of the **Fillet** dialog box, shown in Figure 6-3. First, specify a fillet or round radius by highlighting the default radius and entering a value, or by selecting a value from the list. Then, pick the **Tangent Fillet** or **Smooth (G2) Fillet** button from the flyout to produce the desired fillet or round transition. A tangent fillet is acceptable for most applications and produces a curve that is tangent to adjacent faces. A **Smooth (G2) Fillet** can be selected to create a G2 curve, or curvature-continuous situation, between the curve and adjacent faces.

Next, identify which objects you want to fillet or round. Pick the appropriate **Select Mode** radio button to help with selecting single edges, a closed loop, or an entire feature.

Figure 6-8.
Compare the constant fillets and rounds with the variable fillets and rounds.



Once you choose a select mode, pick the object you want to fillet or round. A preview of the feature is displayed. If you want to fillet all the interior part edges using the same style and radius, pick the **All Fillets** check box. Similarly, if you want to round all exterior part edges with the same style and radius, pick the **All Rounds** check box. When you pick these check boxes, all available feature edges are automatically selected.

NOTE

You can deselect any selected object by holding down the [Ctrl] key and picking the items you want to remove from the operation.

If the preview looks correct and you do not want to place additional fillets or rounds, pick the **OK** button to accept the curves. If you want to apply more constant fillets or rounds, pick the **Click here to add** button. Then, enter the desired fillet or round radius and select the edge. This is the technique required to produce constant fillets and rounds of different radii, using a single **Fillet** tool. Continue adding fillets and rounds as required. When finished, pick the **OK** button.

NOTE

To delete a fillet or round edge listed in the **Edges** list box, pick the edge you want to remove and press the [Del] key. To add variable fillets or rounds and setbacks, in addition to constant fillets and rounds, pick the **Variable** tab.

Exercise 6-1

Complete the Exercise on the Student CD.

Forming Variable Fillets and Rounds

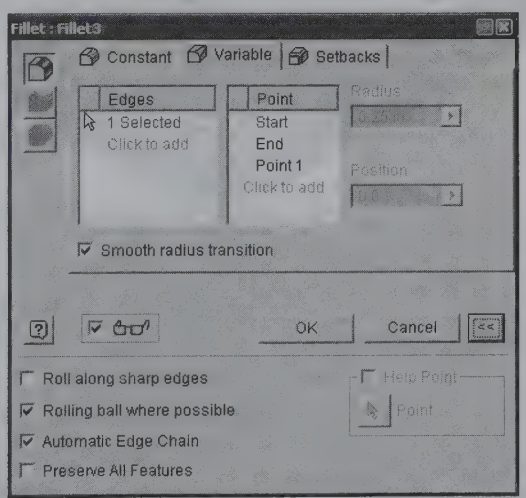
To apply a variable fillet or round, use the options available inside the **Variable** tab of the **Fillet** dialog box, **Edge Fillet** option. See Figure 6-9. Variable fillets and rounds are created by specifying an edge, different radii, and the location of the radii along the edge. To place a variable fillet or round, first select a feature edge. The start and end points are automatically selected, unless you pick a closed loop, such as a cylinder edge, in which case you must pick points. Next, choose additional points along the specified edge by moving your cursor to the desired location. A yellow dot and preview identifies the location of the new point. See Figure 6-10. When you are satisfied with the location of the radius, pick the point. Once all variable round or fillet points are selected, pick a point in the **Point** list box. The selected point is identified by a blue dot. Specify the radius of the point by entering or selecting a value using the **Radius** edit box, and precisely define the location of the point by entering or select a value in the **Position** edit box. Now, continue the process of selecting points in the **Point** list box and setting the radius and position of each point. Remember, the active point displays a blue dot on the selected edge. When finished, pick the **OK** button to create the fillets or rounds.

NOTE



To delete a variable fillet or round edge or point listed in the **Edges** or **Point** list boxes, pick the edge or point you want to remove and press the [Del] key on your keyboard. The position of additional points is relative to the start point of the variable fillet or round, and the individual edge pieces, not a tangent loop.

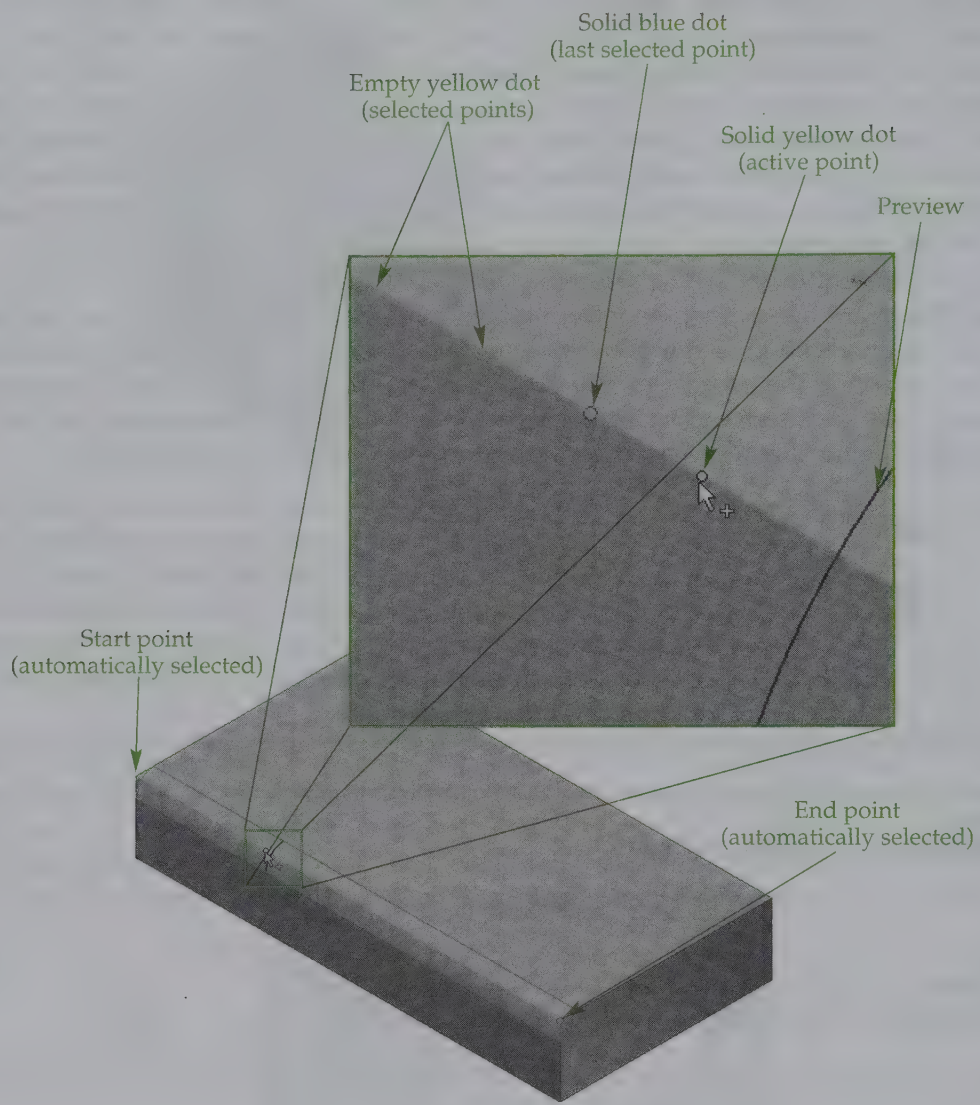
Figure 6-9.
The **Variable** tab of the **Fillet** dialog box.



By default, the **Smooth radius transition** check box is selected. This option creates a steadily increasing or decreasing fillet or round blend between variable points. If you want a direct, linear arc between variable points, deselect the **Smooth radius transition** check box.

If you deselect the **Rolling ball where possible** check box fillet and round corners are blended together. The blend of a non-rolling ball corner is defined by setbacks. *Setbacks* are points where a fillet or round on one edge begins to combine with a fillet

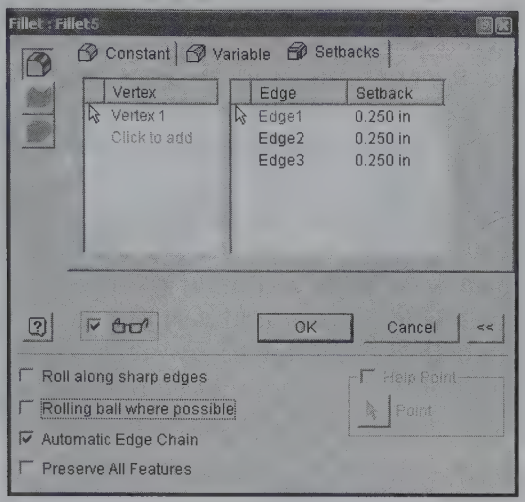
Figure 6-10.
Notice the point style when selecting points for a variable-radius fillet or round.



or round of at least two other edges. Setbacks can be any value, but only a setback that is greater than the fillet or round radius blends corners. A setback that is less than or equal to the fillet or round radius creates a rolling ball effect.

To modify the setbacks of a fillet or round pick the **Setbacks** tab of the **Fillet** dialog box. See Figure 6-11. Next, pick the corner, or vertex, of the feature edges. The first vertex is identified as Vertex1 in the **Vertex** list box, and once selected, displays a blue dot. See Figure 6-12. Once the vertex is specified, define the setback sizes. Each of the edges intersecting the selected vertex is listed in the **Edge** list box. If you pick one of the edge names, the matching edge on the feature is highlighted. If you pick one of the edges on the feature, the corresponding edge name in the **Edge** list box is identified by a cursor to the left of the name. Specify the setback for Edge1 by highlighting the default setback and entering or selecting a value using the **Setback** edit box. Then, pick Edge2 and specify the desired setback. Finally, pick Edge3 and specify the desired setback. If you want to modify additional setbacks, choose another vertex by picking the **Click here to add** button and using the techniques previously described. When complete, pick the **OK** button.

Figure 6-11.
The **Setbacks** tab of the **Fillet** dialog box.



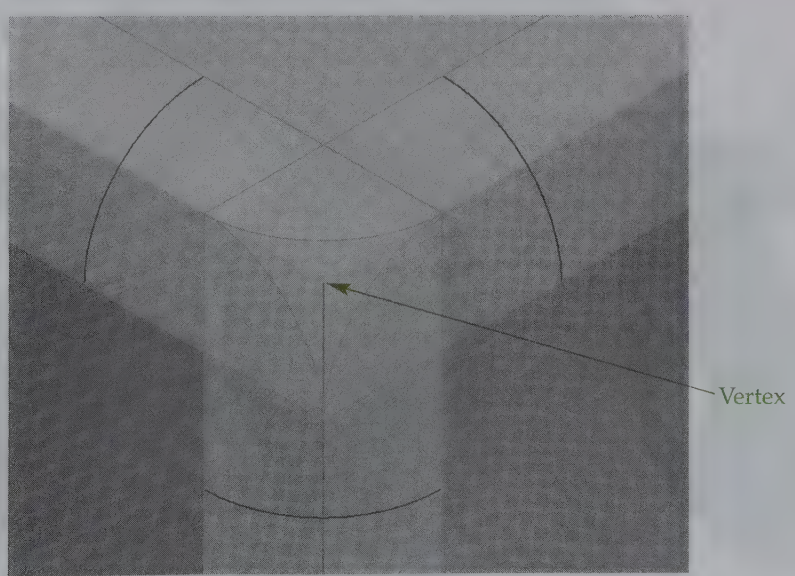
NOTE

To delete a setback vertex or edge listed in the **Vertex** or **Edge** list boxes, pick the vertex or edge you want to remove and press the [Del] key.

Exercise 6-2

Complete the Exercise on the Student CD.

Figure 6-12.
An example of a selected vertex.



Placing Face Fillets and Rounds

Pick the **Face Fillet** button in the **Fillet** dialog box to create a fillet or round according to selected feature faces. The **Face** tab of the **Fillet** dialog box, shown in Figure 6-13, can be used to add a fillet or round to intersecting or nonintersecting feature faces. See Figure 6-14. To apply a face fillet or round, pick the **Face Set 1** button if it is not currently active, and choose the first face used to define the fillet or round. Next, if the **Optimize for Single Selection** check box is selected, the **Face Set 2** button should become active after you pick the first face, allowing you to choose the second face.

Once both faces are selected, specify the fillet or round radius by highlighting the default radius and entering a value, or by selecting a value from the list. A preview will appear if you enter an acceptable radius. If the preview does not look correct, you may be able to adjust the fillet or round by selecting the **Help Point** check box in the **More** area of the **Fillet** dialog box, followed by picking a location on the feature to add the help point. A help point can sometimes be used to help define the transition between the curve and feature faces. If the preview looks correct, pick the **OK** button to accept the curve.

Figure 6-13.
The **Face** tab of the **Fillet** dialog box is used to create face fillets and rounds.

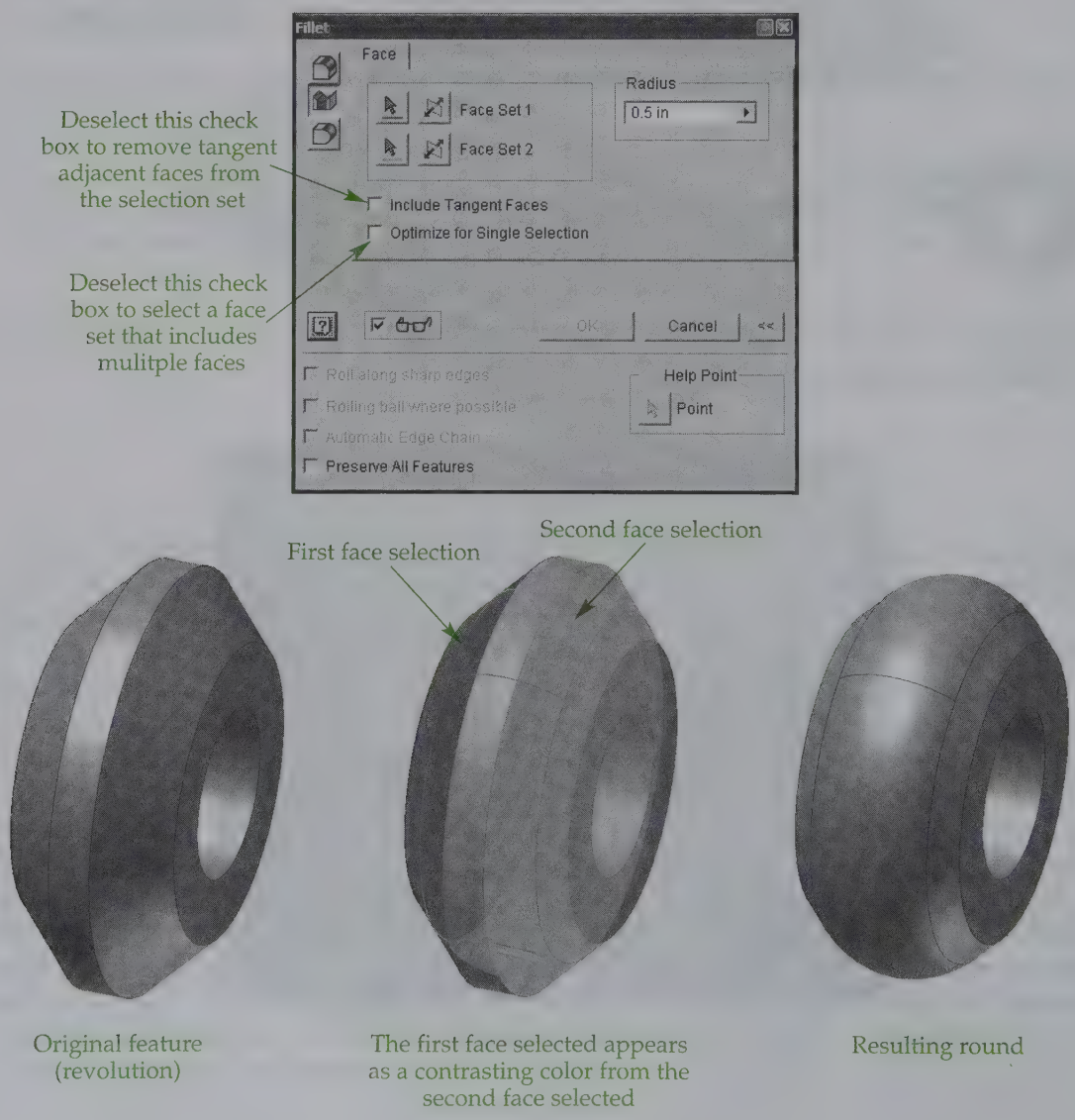
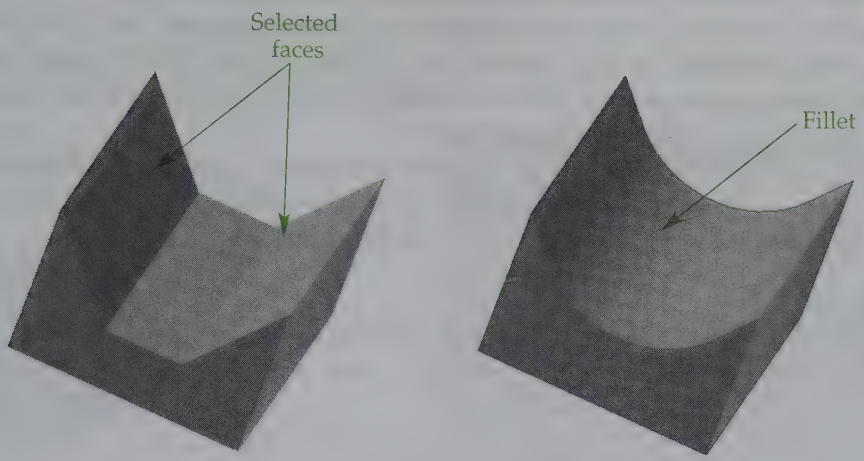
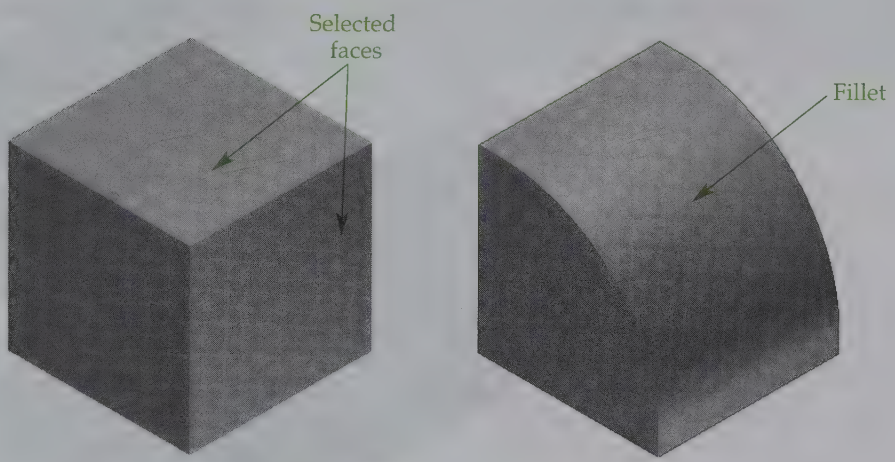


Figure 6-14.
Using the **Face Fillet** option of the **Fillets** dialog box to form fillets and rounds.



Forming a fillet between two nonintersecting feature faces



Forming a fillet between two intersecting feature faces

NOTE

If necessary, use the face set **Flip** buttons to flip the direction of the fillet or round when adding a face fillet or round between surfaces.

Exercise 6-3

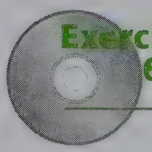
Complete the Exercise on the Student CD.

Adding Full Radius Fillets and Rounds

Full radius fillets and rounds are controlled by the linear dimension of a feature, such as the thickness of a part or width of a slot, and produce half of a circle or cylinder. A full radius fillet or round is created using **Full Round Fillet** option, which forms a full radius according to three selected faces. The full round fillet type is excellent for producing true full radius fillets and rounds that can adapt to design changes. See Figure 6-15. Pick the

Full Round Fillet button in the **Fillet** dialog box to display the **FullRound** tab, used to add a full radius fillet or round. See Figure 6-16. To apply a full radius fillet or round, pick the **Side Face Set 1** button if it is not currently active, and choose the first side face used to define the fillet or round. Next, if the **Optimize for Single Selection** check box is selected, the **Center Face Set** button should become active after you pick the first side face, allowing you to choose the center face. Then, the **Side Face Set 2** button should become active, allowing you to choose the second side face.

Once all sides are selected in an appropriate order, a preview will appear. If the preview looks correct, pick the **OK** button to accept the curve. Figure 6-17 shows a hinge completed by three full rounds and one full fillet.



Exercise 6-4 Complete the Exercise on the Student CD.

Figure 6-15.
Using the **Full round** fillet type to create a true full round that can adapt to design changes.

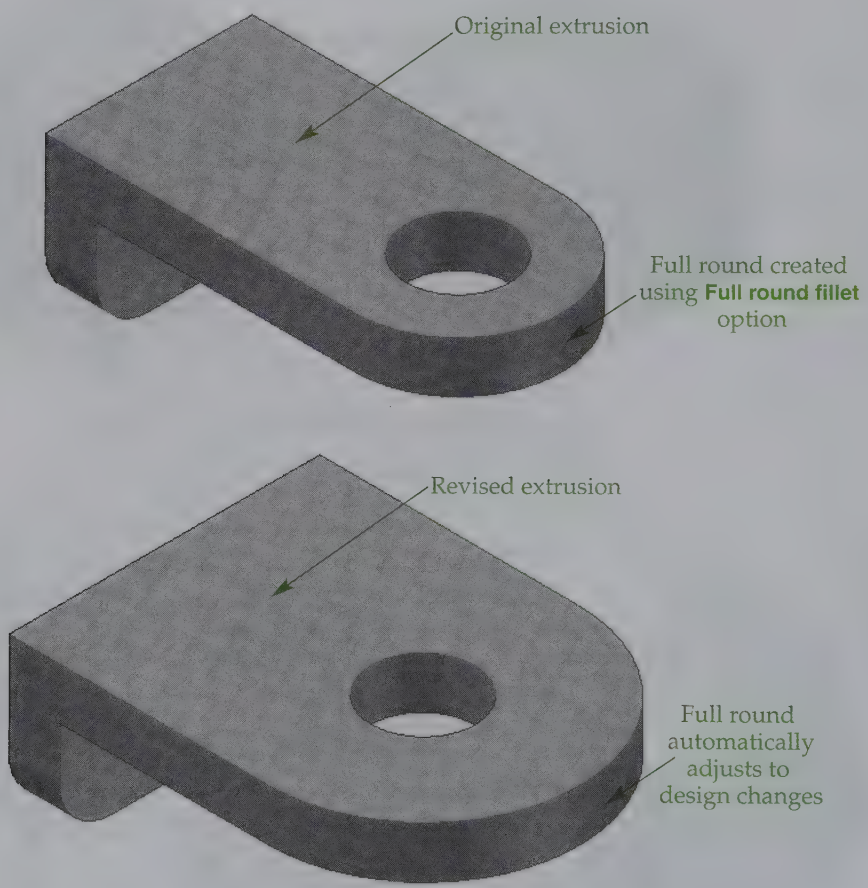


Figure 6-16.
The **FullRound** tab of the **Fillet** dialog box is used to effectively create full radius fillets and rounds.

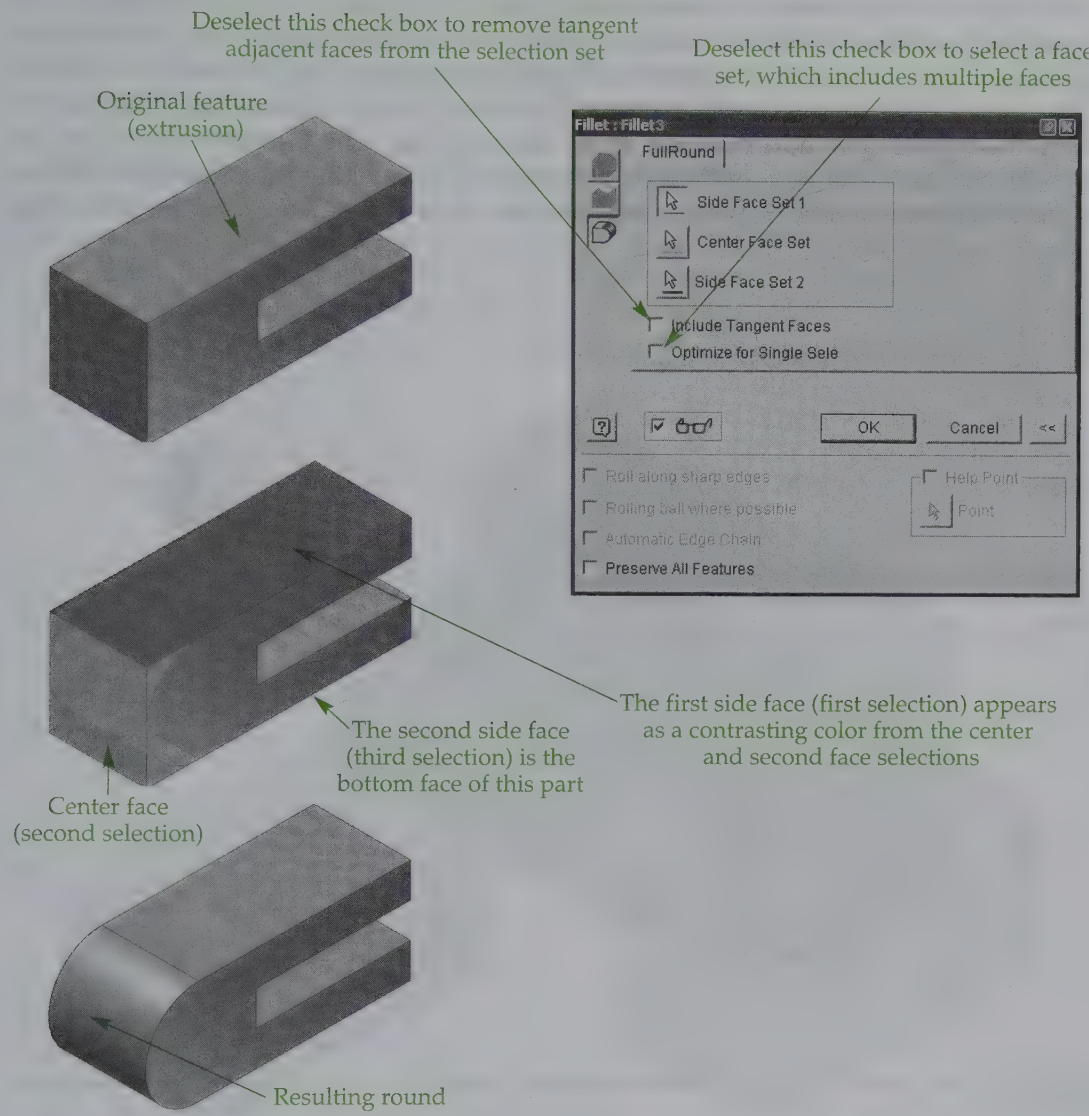
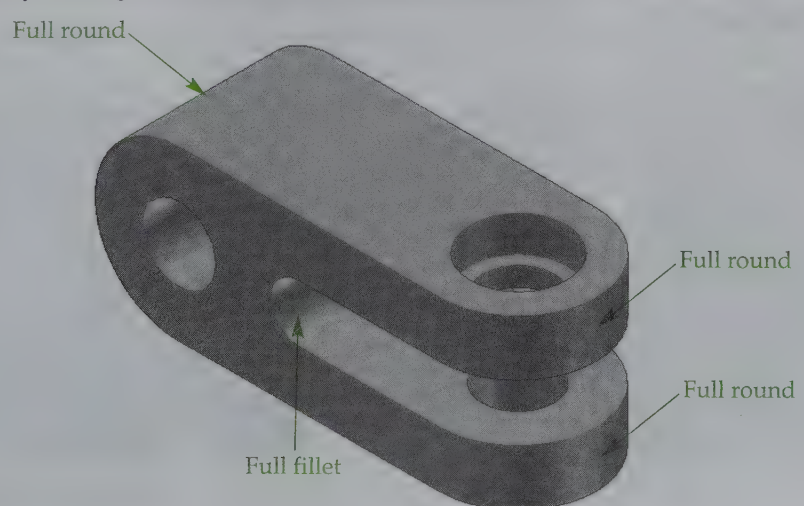


Figure 6-17.
A hinge created by adding three full rounds, one full fillet, and two holes to an extrusion.



Adding Chamfers

A *chamfer* is an angled planar face, placed on a feature edge. Chamfers cut through, or remove, the corner of two existing, intersecting, nonparallel planar faces. See Figure 6-18. Chamfers are used to relieve sharp edges, aid the entry of a pin or thread into a hole, or ease inside corners. Chamfers usually remove material from an external edge, but can add material to an internal edge. The **Chamfer** tool is used to access the **Chamfer** dialog box shown in Figure 6-19, by picking the **Chamfer** button on the **Part Features** panel bar; picking the **Chamfer** button on the **Part Features** toolbar; or pressing [Shift]+[K].

Figure 6-18.
Examples of several chamfers placed on two parts.

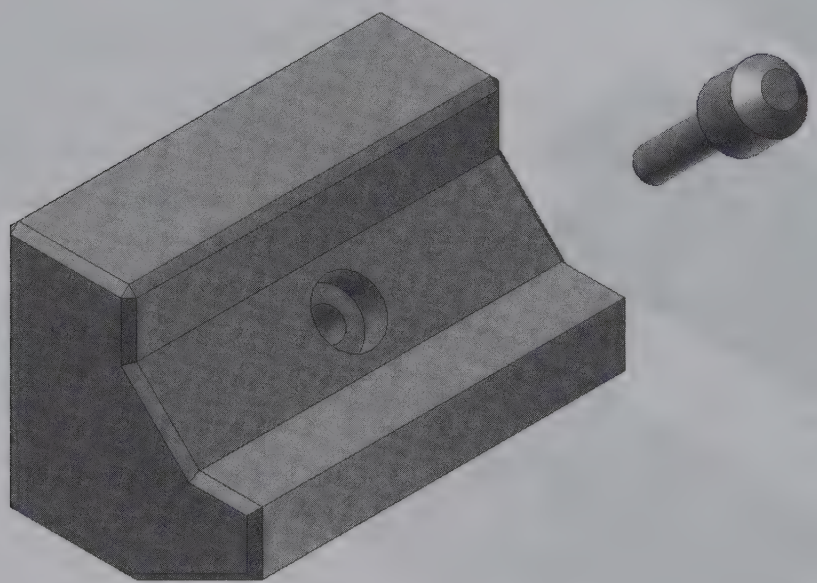
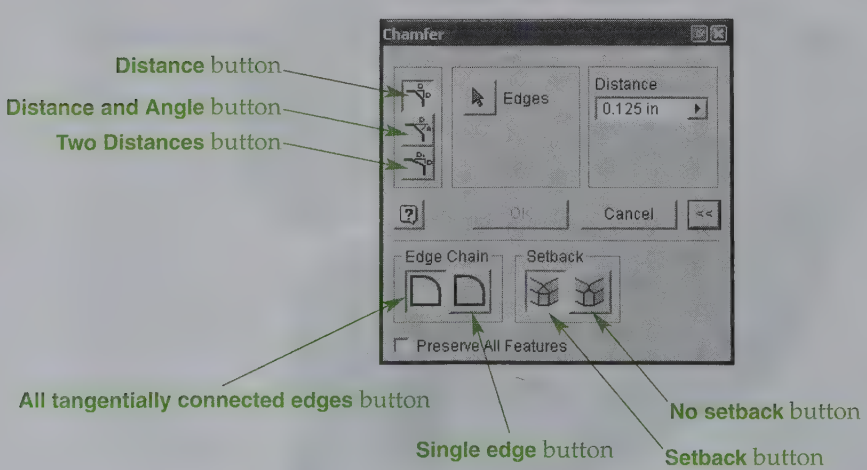


Figure 6-19.
The **Chamfer** dialog box, **Distance** option.



NOTE

You must access the **Chamfer** tool every time you want to create a different size chamfer. However, you can fully chamfer a part using only one **Chamfer** tool if all chamfers are the same. Each chamfer placed during one operation creates a single feature in the browser.

Before you begin to create a chamfer, you may want to adjust the options located in the **More** area of the **Chamfer** dialog box. The following buttons are available when you pick the **More** button to access the **More** area:

- **Edge Chain.** This area allows you to set the type of edge selection. Pick the **Automatic Edge Chain** button to select the entire edge chain of a tangent loop. Pick the **Single Edge** button to select an individual segment of a tangent loop edge. See Figure 6-20.
- **Setback.** This area allows you to define the chamfer corner style when chamfering parts with three or more intersecting edges. See Figure 6-21. Pick the **Setback** button to apply a setback to the intersecting chamfer edges. Pick the **No Setback**

Figure 6-20.
You can select edges for a chamfer by selecting an entire edge chain or an individual edge piece.

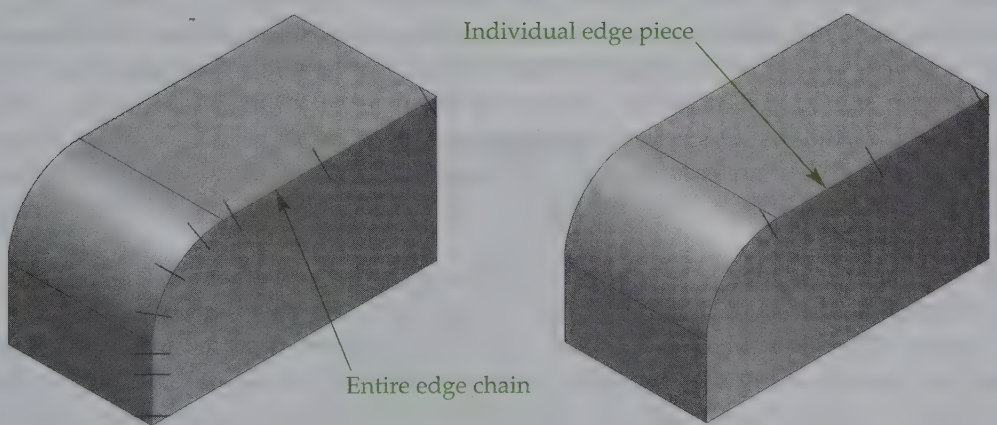
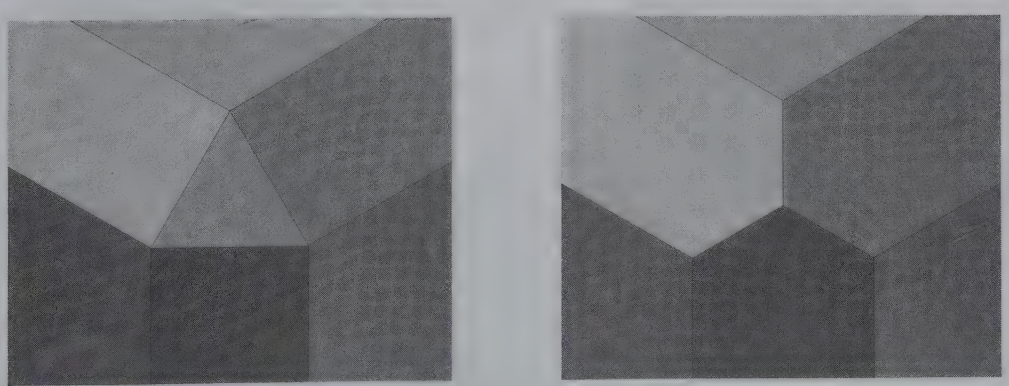


Figure 6-21.
An example of a chamfered corner with the **Setback** button selected and with the **No Setback** button selected.



Setback button selected

No setback button selected

button if you do not want to create a setback. This option establishes a chamfer point at the intersection of the edges instead of a flat surface.

- **Preserve All Features.** Pick this check box when features intersect a corner that you want to chamfer. When the **Preserve All Features** check box is selected, existing feature geometry is not modified by a new chamfer that intersects the feature.

To create a chamfer, first select one of the following chamfer method buttons: **Distance**, **Distance and Angle**, or **Two Distances**. The technique used by each of these buttons to create chamfers is shown in Figure 6-22. The m ethod button selected by default is **Distance**, which is the simplest chamfer option. The **Distance** method allows you to create a chamfer with equal-distance sides from the selected edge, or a 45° angle. To place a chamfer using this option, you must first be sure the **Distance** button is selected. Then pick the edges you want to chamfer and enter or select a chamfer distance using the **Distance** edit box. If the chamfer preview looks correct, pick the **OK** button to create the chamfer.

NOTE



You can deselect any selected edge by holding down the [Ctrl] key and picking the edges you want to remove from the operation.

If you want to specify one chamfer side as a distance from the edge and the other side as an angle from the edge, pick the **Distance and Angle** method button. Figure 6-23 shows the **Chamfer** dialog box with the **Distance and Angle** button selected. Initially, the **Face** button should be active, allowing you to select the face from which you want the chamfer distance from the edge to be measured. Once you define the face, the **Edge** button should become active. The **Edge** button allows you to select the edges from which you want the angle to be measured. To change the face, pick the **Face** button again and then reselect the edges. Once you establish the face and edges you want to chamfer, enter or select a chamfer distance in the **Distance** edit box and enter or select a chamfer angle in the **Angle** edit box. If the chamfer preview looks correct, pick the **OK** button to create the chamfer.

If you want to specify one chamfer side with a distance from the edge, and the other side with a different distance from the edge, pick the **Two Distances** method button. Figure 6-24 shows the **Chamfer** dialog box with the **Two Distances** button selected. To place a chamfer using the **Two Distances** method, pick the edge you want to chamfer. Enter or select a chamfer distance in the **Distance1** edit box, and enter or select a

Figure 6-22.
Examples of the effects of each type of chamfer creation method.

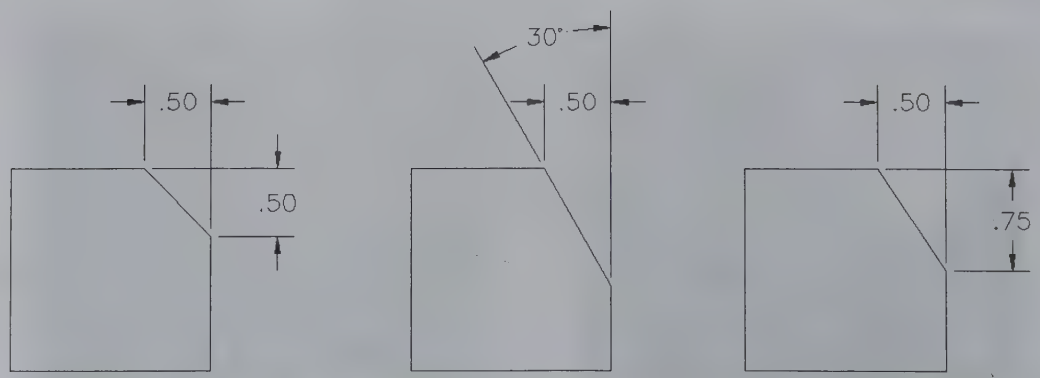


Figure 6-23.
The angle and distance of the chamfer are in the list boxes of the **Chamfer** dialog box and drawn on the part.

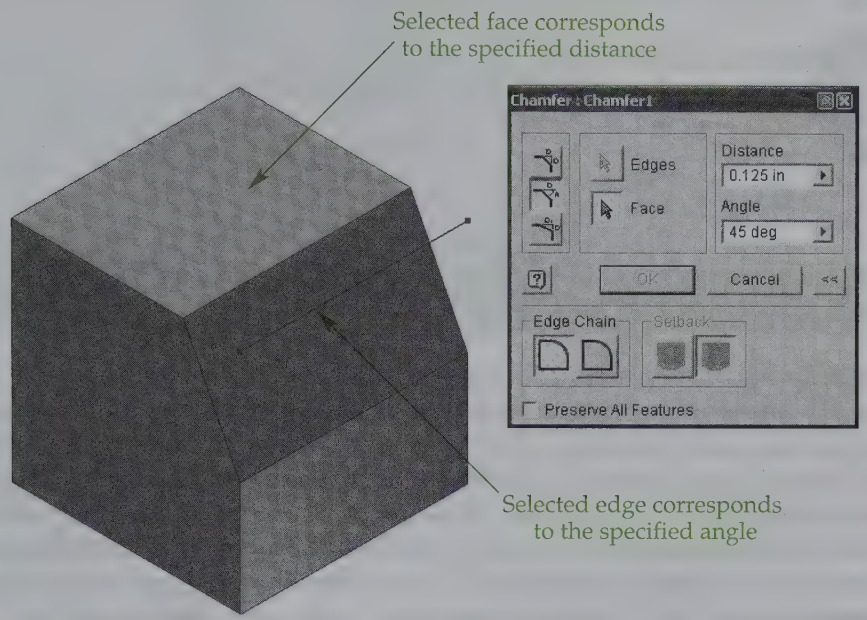
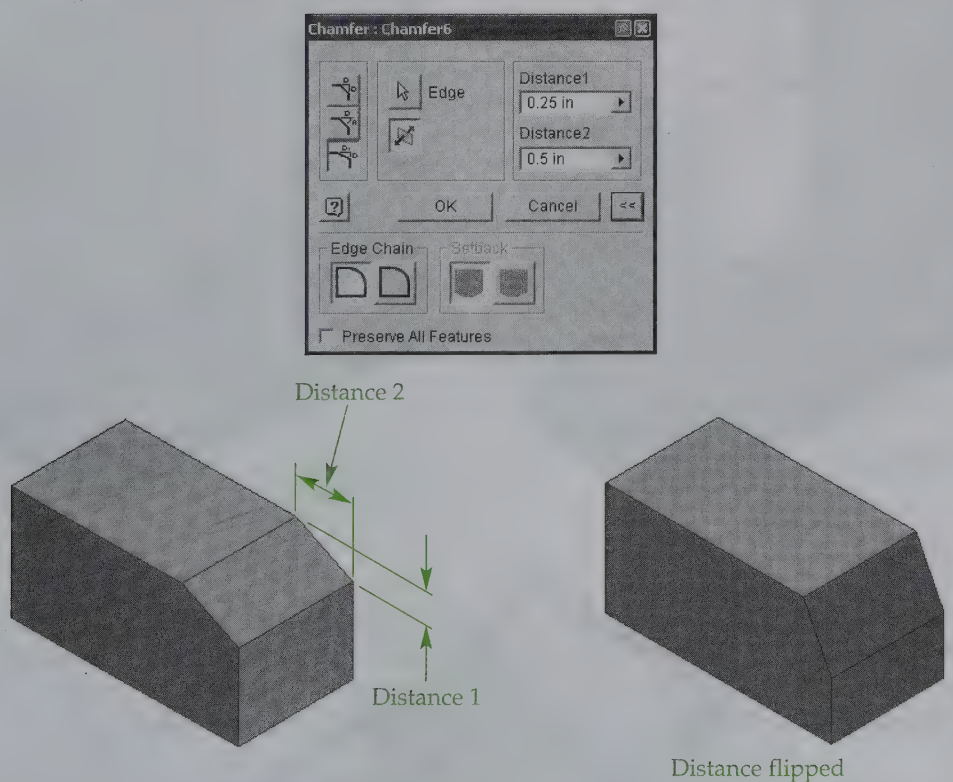


Figure 6-24.
The distances of the chamfer are in the list boxes of the **Chamfer** dialog box and drawn on the part. Notice the position of the chamfer when the **Flip Direction** button is pressed.



chamfer distance in the **Distance2** edit box. If the chamfer preview looks correct, pick the **OK** button to create the chamfer. However, if the distances you enter need to be reversed, pick the **Flip** button to redefine the direction of the chamfer distances from the edge. Pick the **OK** button to create the chamfer.



Complete the Exercise on the Student CD.

Shelling Features

A ***shell*** removes material from a feature and creates a hollow space or opening. See Figure 6-25. Shells are typically used to design parts for casting or forging. However, shells can be used for any application where it is necessary to remove material from a feature, leaving behind a specified wall thickness. The material removed by a shell is determined by one or more selected feature faces. Shells modify features by removing material and creating new faces on the inside, outside, or inside and outside of existing feature faces. Shell features are created using the **Shell** tool. To access the **Shell** tool, pick the **Shell** button on the **Part Features** panel bar, or pick the **Shell** button on the **Part Features** toolbar.

The **Shell** dialog box is displayed when you access the **Shell** tool. See Figure 6-26. To place a shell, first pick the **Remove Faces** button, if it is not already selected. The **Remove Faces** button allows you to specify which feature faces you want to remove during the shelling process. You can remove one or more feature faces. See Figure 6-27. Once you specify the faces you want to remove, determine which direction to shell

Figure 6-25.
An example of a shelled part.

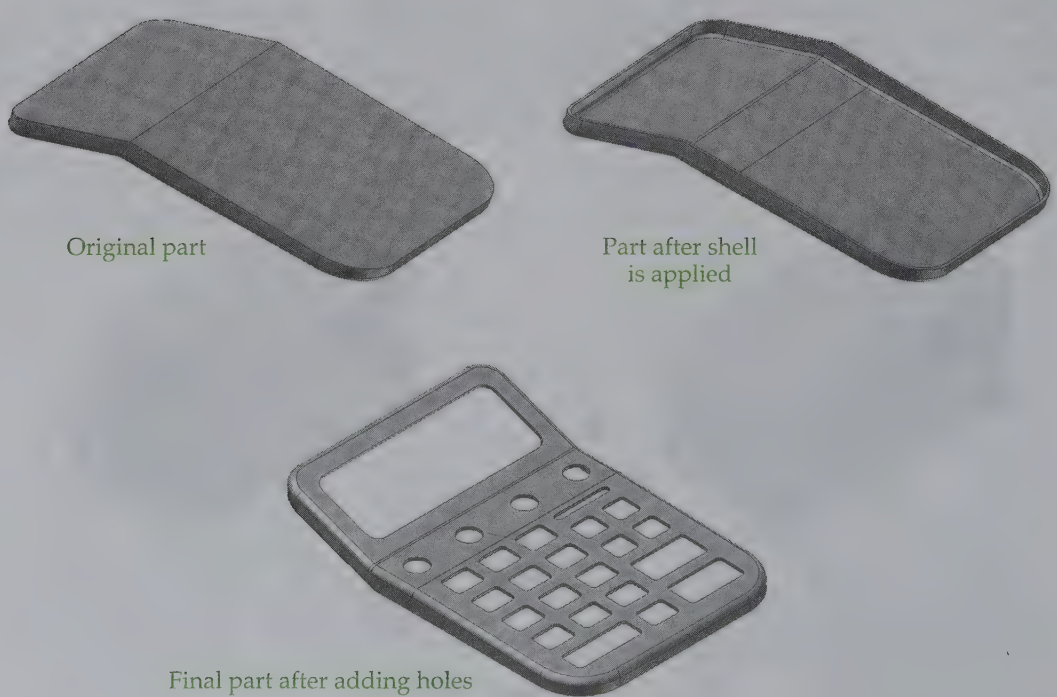


Figure 6-26.
The **Shell** dialog box.

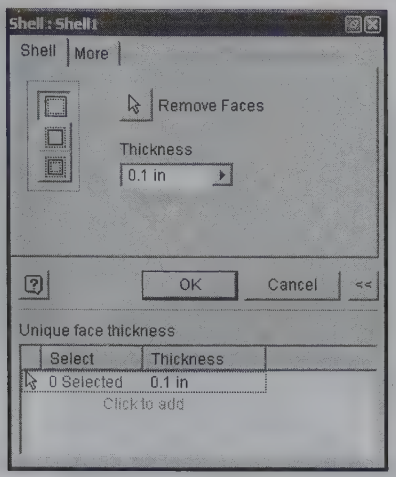
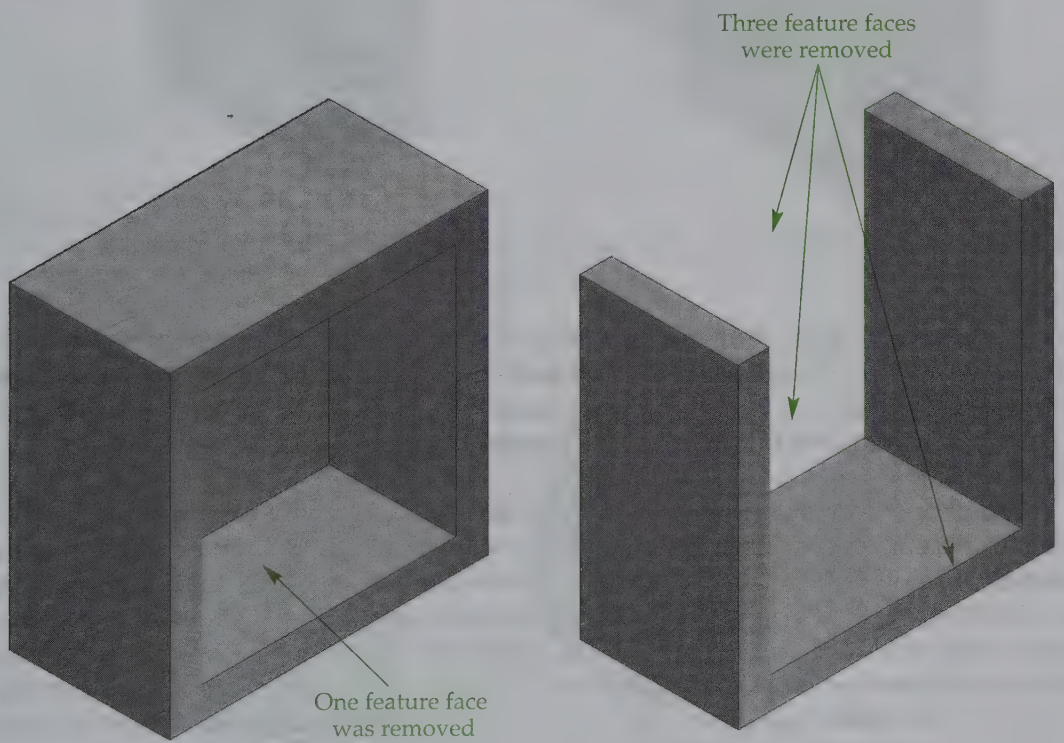


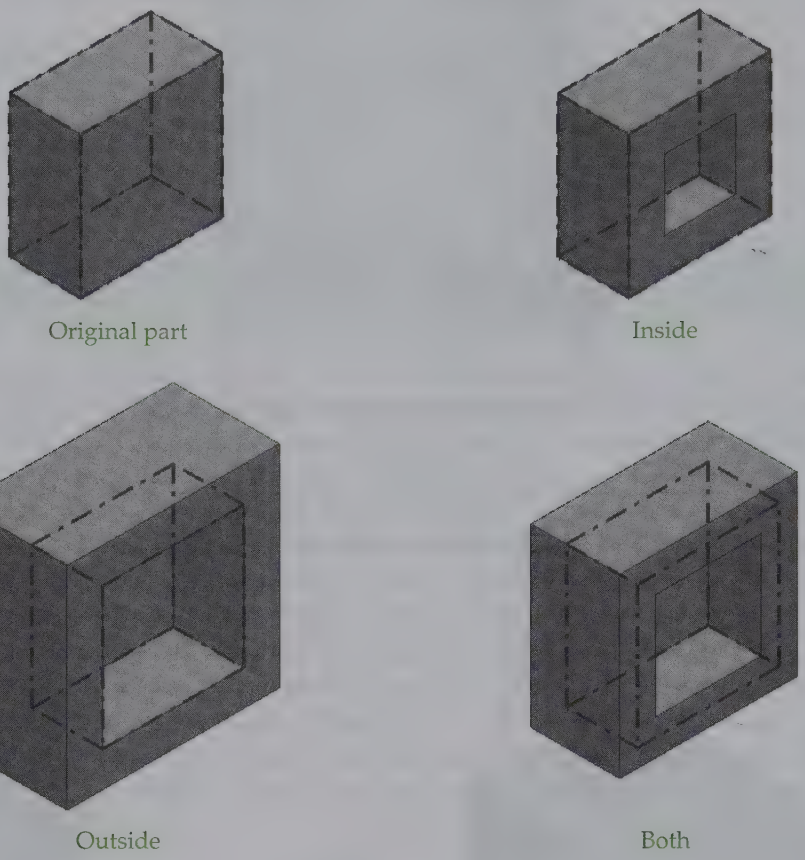
Figure 6-27.
Removing one feature face and multiple feature faces during a shell operation.



the feature. Selecting the **Inside** button allows you to create shell walls on the inside of the existing feature faces. Picking the **Outside** button allows you to create shell walls on the outside of the existing feature faces. Selecting the **Both** button allows you to create shell walls equally thick on the inside and the outside of the existing feature faces. See Figure 6-28.

The next step in creating a shell feature is defining the wall thickness, which determines the amount of material left after the shell operation. To define and apply a uniform thickness throughout the shell, enter or select a value using the **Thickness** edit box. Pick the **OK** button to create the shell.

Figure 6-28.
When shelling a part, the shell thickness can be maintained inside, outside, or inside and outside of the original part.



If you want to apply different thicknesses to certain walls, as shown in Figure 6-29, pick the **More** button to access the **Unique face thickness** area. Pick the **Click here to add** button and select the face where you want to place a unique wall thickness. Finally, enter or select the desired thickness using the **Thickness** edit box and pick the **OK** button to create the shell.

A shell operation forms a new surface that is parallel to the existing surface. The new parallel surface, or offset, is set according to the thickness you enter when creating the shell. The offset is consistent throughout the shell except for situations when the specified thickness is too large. An example can be seen in Figure 6-30, which shows a sharp pointed loft that has been shelled. Notice the shell thickness is not consistent when it becomes too large to form a parallel surface. To adjust for this issue pick the **More** tab in the **Shell** dialog box. See Figure 6-31. The **Allow Approximation** check box is selected by default, and attempts to produce the shell by adjusting the thickness as needed. The following options are selected from the drop-down list to control the approximate solution:

- **Never too thin.** Select this option to increase the shell thickness as necessary, but never less than the specified thickness.
- **Never too thick.** Pick this option to decrease the shell thickness as necessary, but never more than the specified thickness.
- **Mean.** Choose this option to increase or decrease the thickness as needed.

Once you choose the desired approximation option, pick the **Optimized** radio button to use an approximation tolerance that will result in the quickest shell, or choose the **Specify Tolerance** radio button to enter a tolerance. The approximation tolerance you enter is a percentage. The lower the percentage, the higher the approximation accuracy.

Figure 6-29.
Pick the **More** button to add unique face thicknesses to a shell feature.

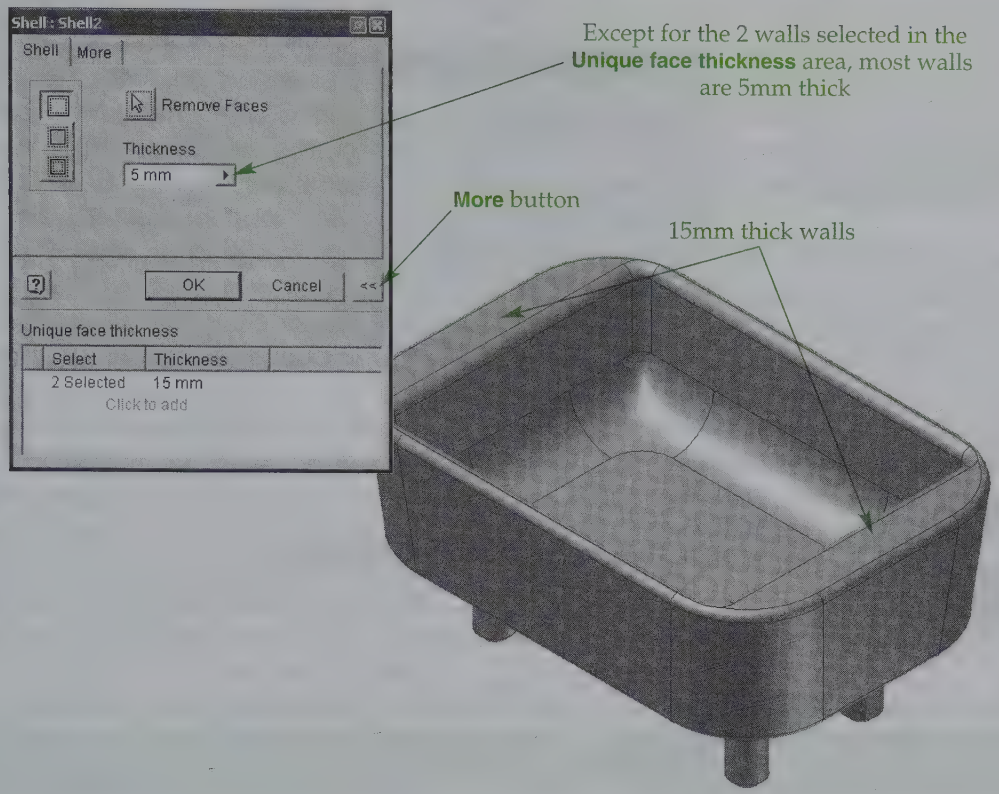
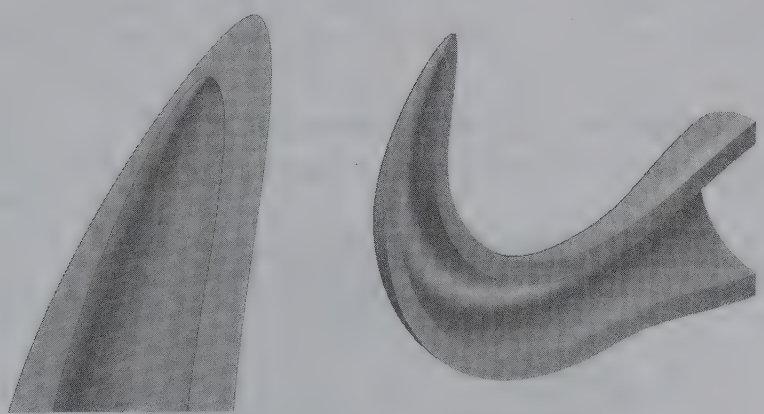


Figure 6-30.
An example of approximating shell offset when a consistent thickness is too large to form a parallel curved surface.

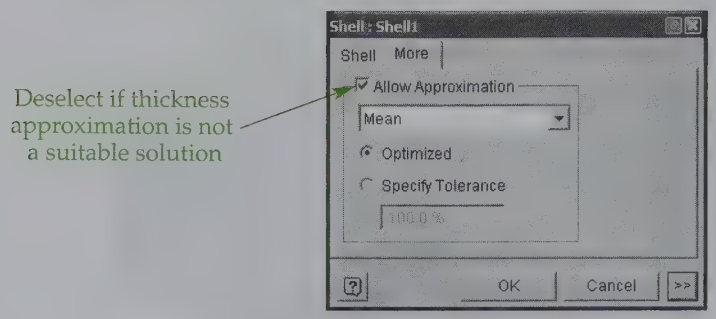


NOTE



If you choose to approximate a shell solution, an alert box will indicate that the offset is approximate. Pick the **Accept** button to produce the shell, or pick the **Edit** button to make modifications to the shell. Deselect the **Allow Approximation** check box if you do not want the shell thickness to be approximated. This will usually require that you make significant changes to the model before creating the shell.

Figure 6-31.
The **More** tab of the **Shell** dialog box is used to approximate shell thickness.



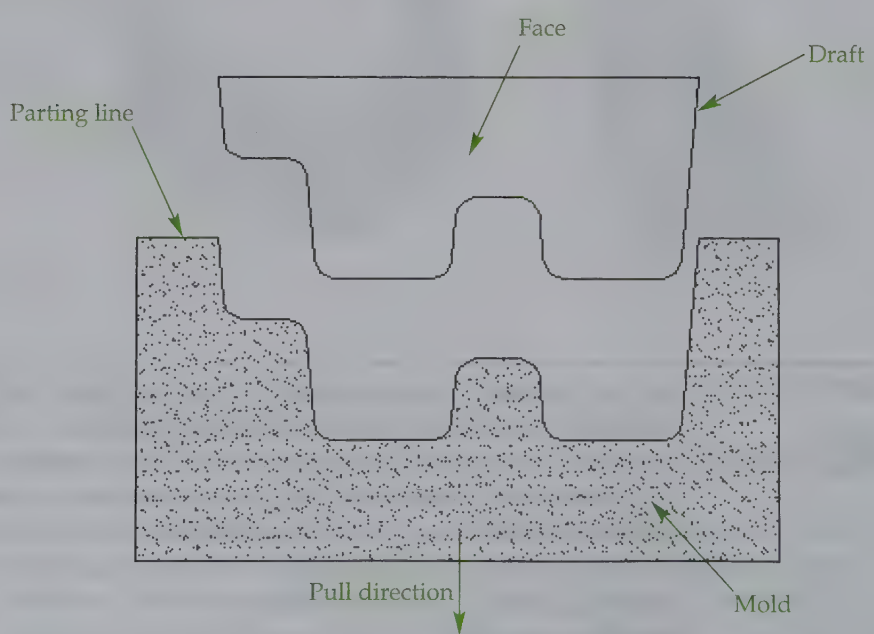


Complete the Exercise on the Student CD.

Applying Face Draft

A ***face draft*** is a taper placed on a part surface, usually to aid in the removal of a part or pattern from a mold. See Figure 6-32. Draft features are created using the **Face Draft** tool. To access the **Face Draft** tool, pick the **Face Draft** button on the **Part Features** panel bar; pick the **Face Draft** button on the **Part Features** toolbar; or press the [Shift]+[D].

Figure 6-32.
An example of when face draft is typically applied to a part.



The **Face Draft** dialog box is displayed when you access the **Face Draft** tool. See Figure 6-33. To draft a face or group of faces, choose the desired draft type button. By default, the **Fixed Edge** button is selected, which allows you to identify the face draft pull-direction and the faces to be drafted using fixed edges. In contrast, selecting the **Fixed Plane** button allows you to define a fixed plane and the faces to be drafted in reference to the fixed plane. The main difference between the **Fixed Edge** and **Fixed Plane** options is that material is added or removed at the selected edge using the **Fixed Edge** function, while material is both added and removed at the selected plane using the **Fixed Plane** function.

To create a face draft using the **Fixed Edge** option, pick the **Pull Direction** button, if it is not already selected, and identify the face draft pull-direction by selecting the desired face or plane. The ***pull direction*** is the direction from which the casting mold is pulled or removed from the part. See Figure 6-32. When you select the pull direction, an arrow identifies which way the mold is pulled. See Figure 6-34. If you want to reverse the specified pull direction, pick the **Flip** button. Once you designate the pull direction, the **Faces** button should automatically become active, allowing you to select the faces you want to draft. When you move your cursor over feature faces, a vector preview of the face draft operation is displayed. The vector indicates the fixed edge and angle of the face draft. Once the vector displays the correct orientation, pick the face. When you identify the pull direction and the faces you want to draft, enter or select the desired draft angle using the **Draft Angle** edit box and pick the **OK** button. Figure 6-34 shows the drafted faces.

To create a face draft using the **Fixed Plane** option, pick the **Draft Plane** button if it is not already selected, and identify the face draft pull direction by selected a face or plane. For most applications the plane you select should be a work plane or an existing feature face that can be used to both add and remove material during the face draft operation. When you choose the fixed plane, an arrow identifies which way the part is pulled. See Figure 6-35. If you want to reverse the specified pull direction, pick the **Flip** button. Once you designate the pull direction, the **Faces** button should automatically become active, allowing you to select the faces you want to draft. When you move your cursor over feature faces, a vector preview of the face draft operation is displayed. See Figure 6-35. The vector indicates the fixed edge and angle of the face draft according to the selected

Figure 6-33.
The **Face Draft** dialog box with the **Fixed Edge** button selected and the **Fixed Plane** button selected.

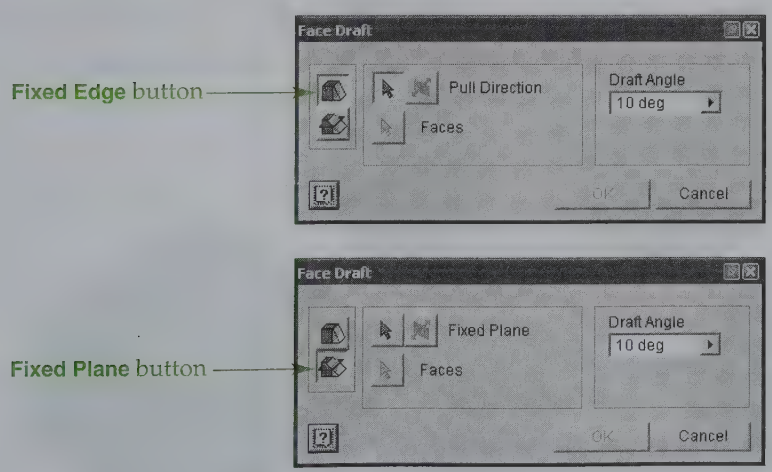


Figure 6-34.
Notice the position of the indicators during a fixed-edge face-draft selection and the resulting part.

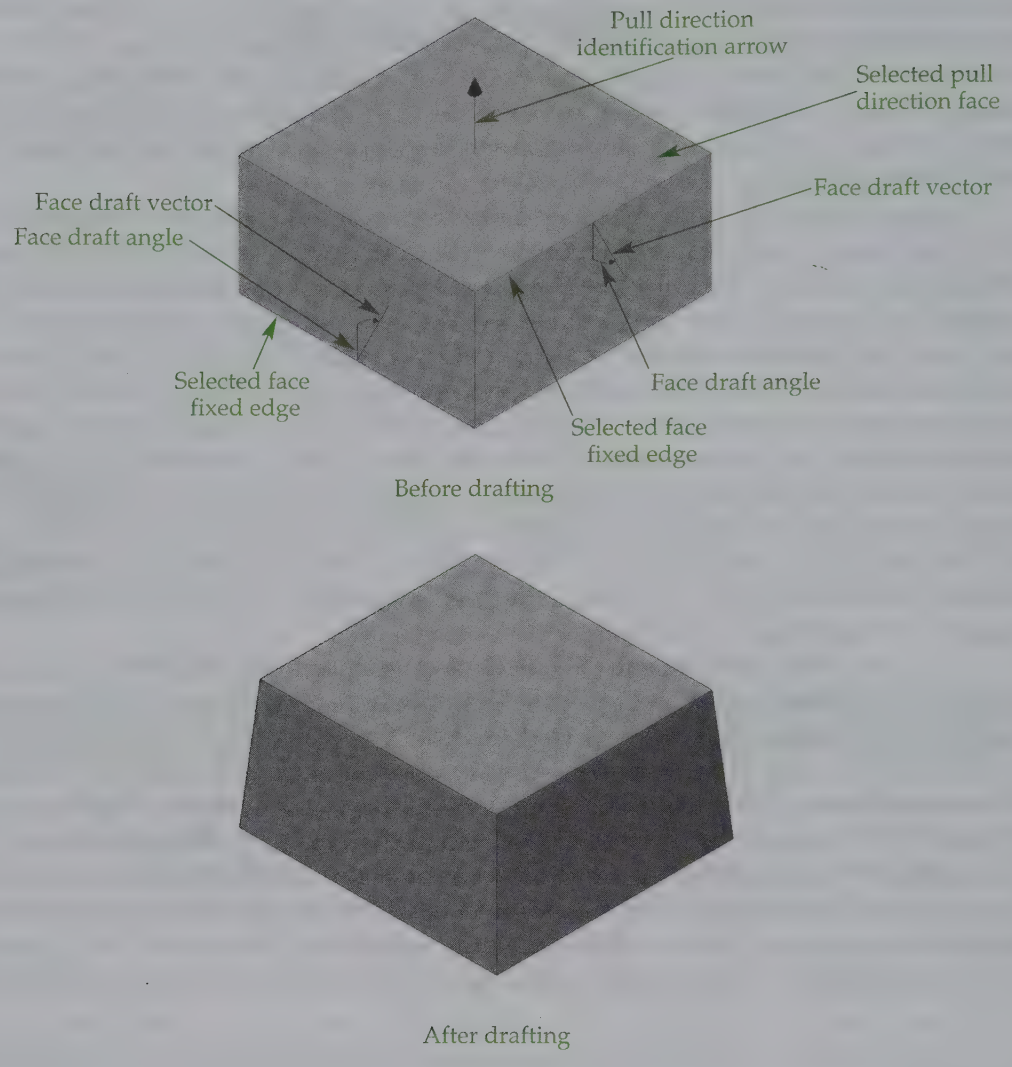
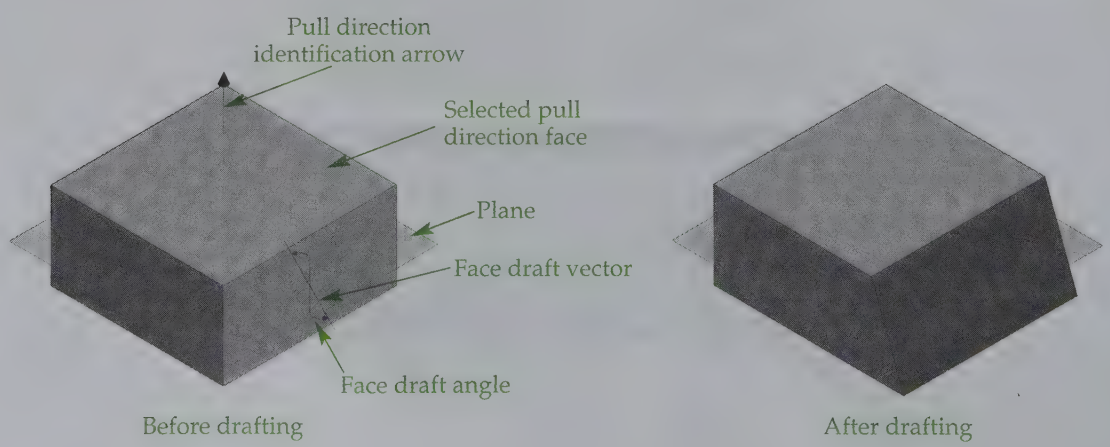


Figure 6-35.
Notice the position of the indicators during a fixed-plane face-draft selection and the resulting part.



fixed plane. Notice you can only define the face draft angle and fixed edges in relation to the selected fixed plane. When you identify the pull direction and the faces you want to draft, enter or select the desired draft angle using the **Draft Angle** edit box and pick the **OK** button. Figure 6-35 shows the faces before and after drafting.

Exercise 6-7 Complete the Exercise on the Student CD.

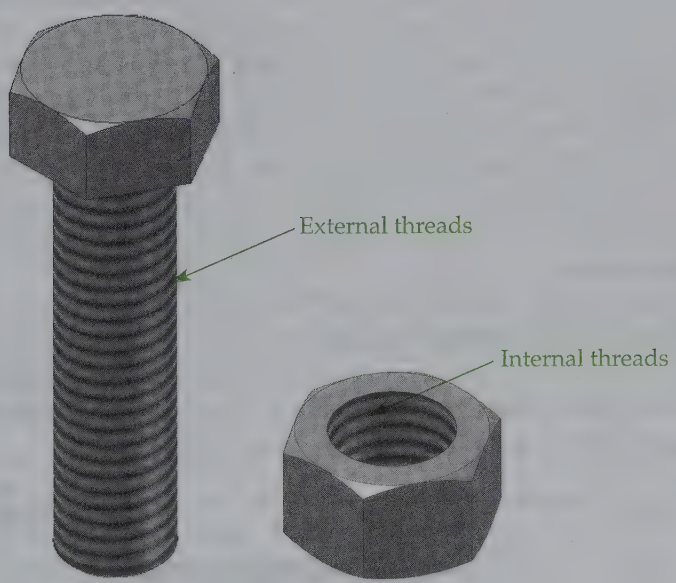
Placing Threads

Threads are grooves cut in a spiral fashion in or around the face of a cylindrical or conical feature. Threads can be placed both externally and internally. ***External threads*** are thread forms on an external feature such as a pin, shaft, bolt, or screw. ***Internal threads*** are thread forms on an internal hole feature. See Figure 6-36.

NOTE

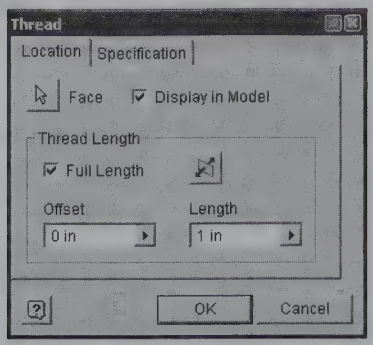
Although the **Thread** tool allows you to add threads to a hole, you may want to use the thread options available in the **Holes** dialog box to create threaded holes.

Figure 6-36.
An example of externally and internally threaded features.



Threads are created using the **Thread** tool, but this tool does not actually cut threads into a feature. It displays a bitmap image of threads on the cylinder, cone, or hole face. Access the **Thread** tool by picking the **Thread** button on the **Part Features** panel bar, or picking the **Thread** button on the **Part Features** toolbar. This opens the **Thread Feature** dialog box shown in Figure 6-37.

Figure 6-37.
The **Thread** dialog box, **Location** tab.



NOTE

Thread representations created using the **Thread** tool are fully parametric, which means the information you use to create the threads is used to annotate drawings and is available for other applications. Thread representations, using the **Thread** tool, are adequate for most applications, but if you need to create actual detailed threads, you must use the **Coil** tool.

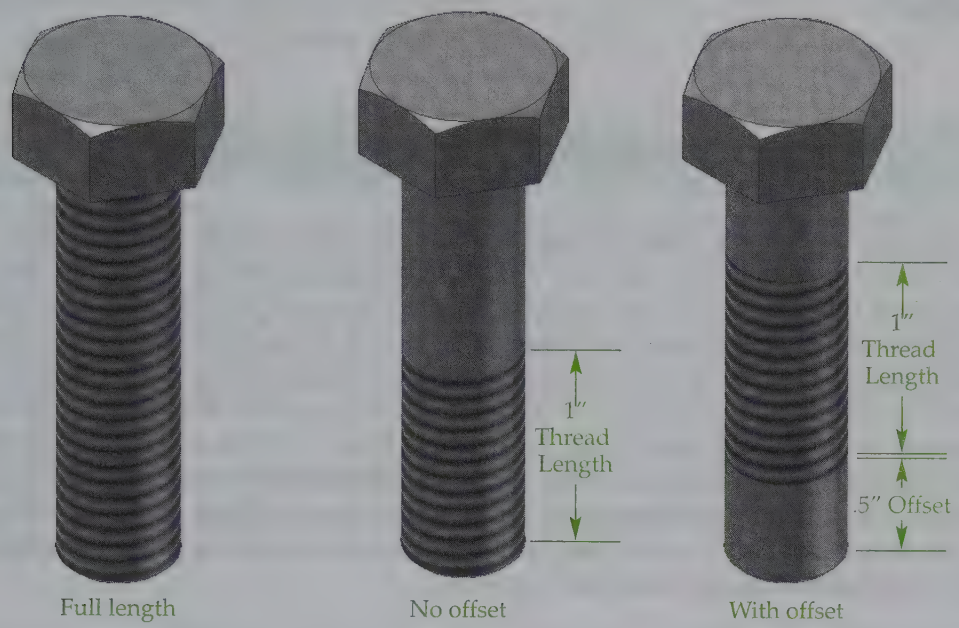
To apply threads to an interior or exterior cylindrical or conical face, pick the **Face** button, if it is not automatically selected. Specify the thread length, using the **Thread Length** area of the **Thread Feature** dialog box. If you want the threads to encompass the entire length of the feature face, pick the **Full Length** check box. If you want to specify a certain length of threads and create threads that do not run the entire length of the face, deselect the **Full Length** check box. Enter or select the desired thread length in the **Length** edit box. Once you specify the thread length, enter or select an offset distance in the **Offset** edit box. *Offset* is the distance from the edge of the face to the beginning of threads. See Figure 6-38. Once you specify the thread length and offset, decide if you want to hide the bitmap thread representation on the part. By default the thread representation is displayed. However, if you want to hide the image, deselect the **Display in Model** check box.

When you define all the thread location options in the **Location** tab, pick the **Specification** tab of the **Thread Feature** dialog box to identify the thread properties. See Figure 6-39. The thread characteristics available when placing threads are the same as those offered when creating holes. Refer to the Creating Holes section of Chapter 5 for more information regarding thread characteristics.

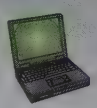
NOTE

Specific thread types, such as NPT, are available when you add threads to a conical feature to replicate tapered pipe threads. Only designation options that correspond to the selected size are available.

Figure 6-38.
Threads may cover the full length of a face, partial length without an offset, or partial length with an offset.

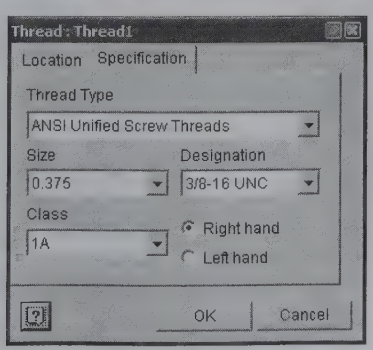


PROFESSIONAL TIP



The thread information available inside the **Specification** tab is outlined by Inventor on a spreadsheet. The location of the spreadsheet is: Program Files\Autodesk\Inventor 11\ Design Data\Thread.xls. Use this spreadsheet to modify and add thread types and thread information for your specific applications. Changes made to the **Thread** spreadsheet do not alter existing thread specifications. For more information regarding adding threads to the **Thread** spreadsheet, refer to the "To add thread data to a spreadsheet" section of the Inventor help files. Once you specify the thread preferences in the **Specification** tab, pick the **OK** button to apply the threads to the feature.

Figure 6-39.
The **Thread** dialog box, **Specification** tab.





Complete the Exercise on the Student CD.

Thickening and Offsetting

In some situations when developing more complex models, you may need to add material to a face or surface or create a construction surface using an existing face or surface. Often, the easiest, only, or most effective way to accomplish these tasks is to thicken or offset feature shapes using the **Thicken/Offset** tool. *Thickening* refers to the process of adding a solid to a face or surface and is similar to a solid extrusion. *Offsetting* in the part feature environment refers to the process of offsetting a surface from a face or surface and is similar to offsetting a work plane from a face. Figure 6-40 shows how relatively complex model shapes can be generated using the **Thicken/Offset** tool, in addition to other tools. Access the **Thicken/Offset** tool to display the **Thicken/Offset** dialog box by picking the **Thicken/Offset** button on the **Part Features** panel bar, or picking the **Thicken/Offset** button on the **Part Features** toolbar. See Figure 6-41.

Figure 6-40.
Examples of **Thicken/Offset** tool applications.

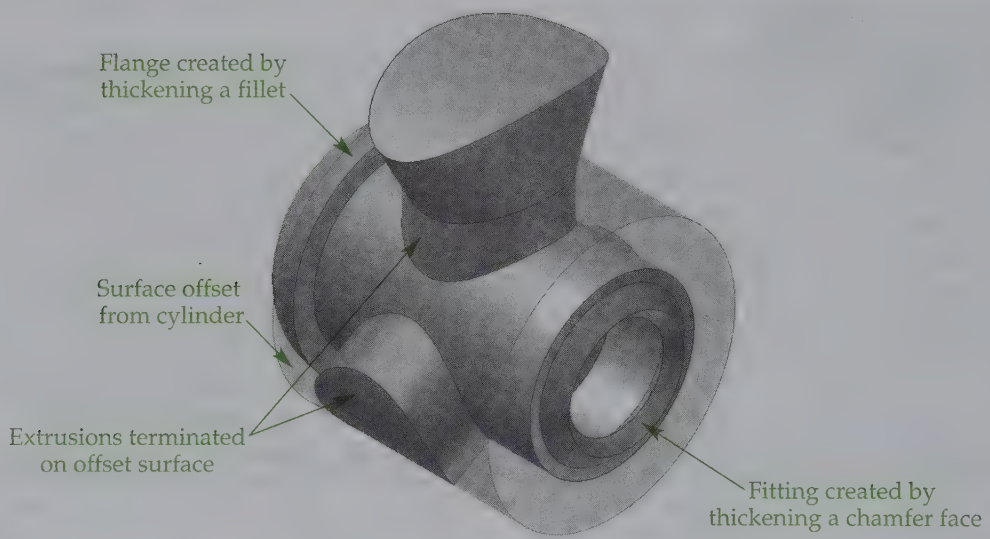
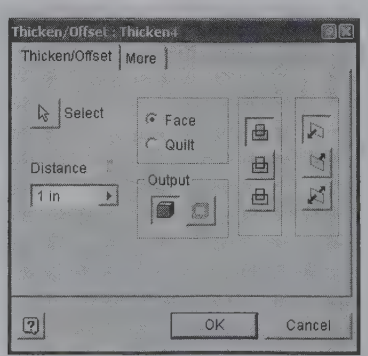


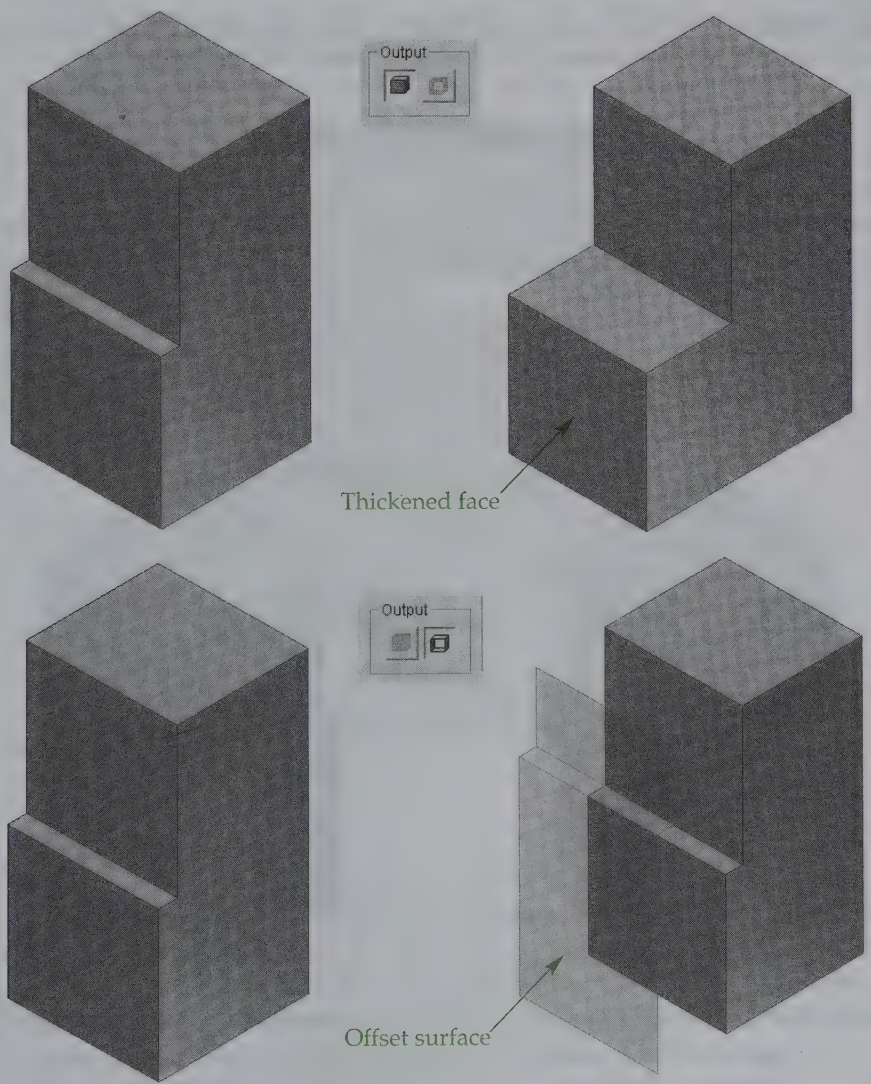
Figure 6-41.
The **Thicken/Offset** dialog box, **Thicken/Offset** tab displayed.



To thicken or offset a face or surface, pick the **Select** button from the **Thicken/Offset** tab if it is not already active. Pick the feature face or surface that you want to thicken or offset. Once the item is selected, use the following options to define the feature:

- **Face.** Though you can choose multiple faces, you cannot pick both faces and surfaces. So, pick the **Face** radio button if you only want to have selection access to faces.
- **Quilt.** Again, though you can choose multiple surfaces, or quilts, you cannot pick both faces and surfaces. So, pick the **Quilt** radio button if you only want to have selection access to surfaces.
- **Output.** This area contains the **Solid** and **Surface** buttons. Pick the **Solid** button to generate a solid thickening, or the **Surface** button to create a volumeless surface offset. See Figure 6-42.
- **Distance.** Use this edit box to enter or select the distance from which the thickened or offset feature will occur in reference to the selected face or surface. You must specify a distance greater than 0 when thickening a feature, but you

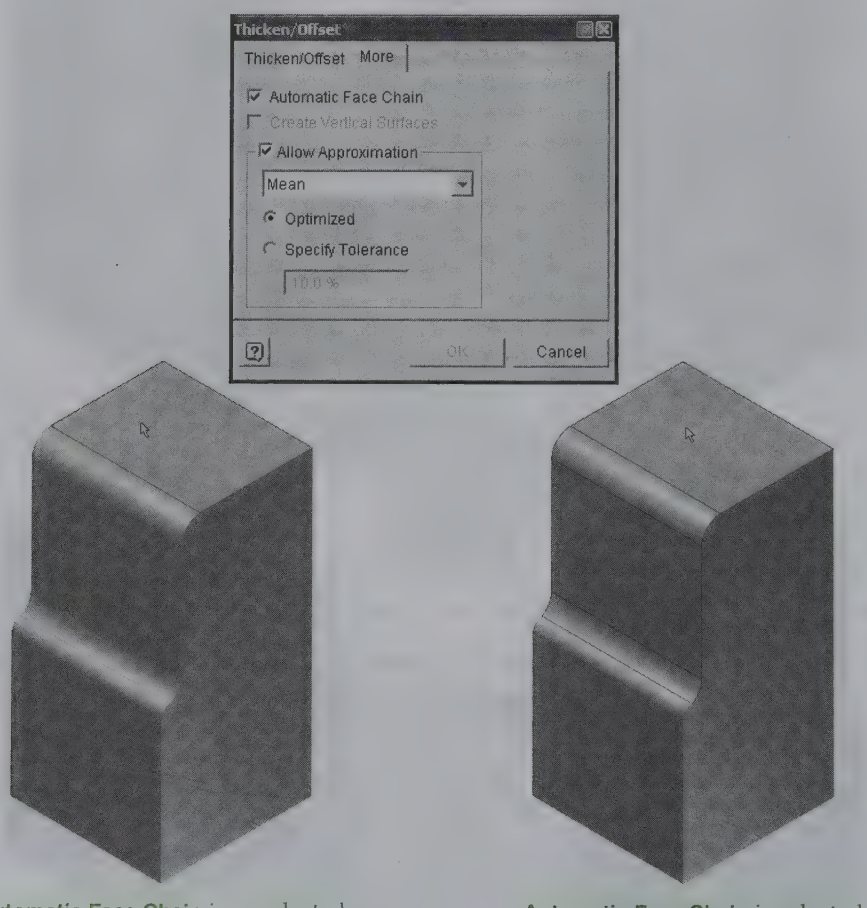
Figure 6-42.
Notice the buttons from the **Output** area of the **Thicken/Offset** dialog box and the resulting geometry.



can enter any distance including 0 when offsetting a surface. Specifying an offset distance of 0 will create a copy of the selected surface.

- **Operation.** These buttons are available only when thickening faces and allow you to choose whether the thicken will join, cut, or intersect another feature.
- **Direction.** These buttons control which side of the selected face or surface is thickened or offset. You can choose to thicken or offset in a positive or negative direction, or thicken or offset equal amounts on both sides of the selection.
- **More.** This tab controls additional thicken or offset characteristics. See Figure 6-43. Pick the **Automatic Face Chain** check box if you want to select multiple connected and tangent faces, instead of having to select each face individually. Pick the **Create Vertical Surfaces** check box if you want to extend surfaces when offsetting external surfaces. The **Allow Approximation** check box is also available in the **More** tab. A thicken or offset operation forms a new surface that is parallel to the existing surface. The new parallel surface is set according to the distance you enter when creating the thicken or offset. The distance is consistent throughout the operation except for situations when the specified distance is too large. The **Allow Approximation** check box is selected by default, and attempts to produce the shell by adjusting the thickness as needed. The approximation options are the same as those described in the Shelling Features section of this chapter.

Figure 6-43.
The **Thicken/Offset** dialog box, **More** tab contains an **Automatic Face Chain** check box that allows you to select multiple connected and tangent faces with one pick.



Automatic Face Chain is unselected

Automatic Face Chain is selected

PROFESSIONAL TIP

If you want to select an entire face chain, you should access the **More** tab of the **Thicken/Offset** dialog box first, pick the **Automatic Face Chain** check box, and then select the faces. Once you specify all the thicken/offset options, pick the **OK** button to generate the feature.

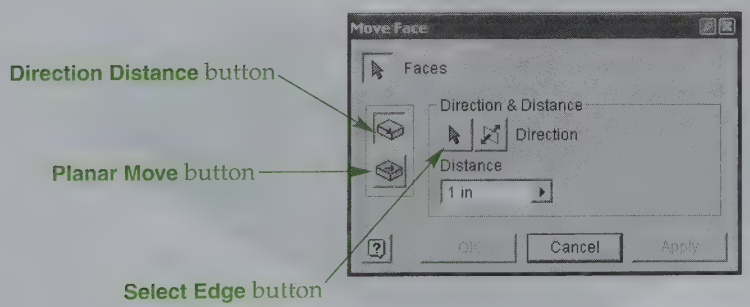
Exercise 6-9

Complete the Exercise on the Student CD.

Moving Faces

If the size or shape of a feature requires modification, you should edit the feature parameters as needed. For example, to change an extrusion from 3″ to 2″, edit the extrusion distance using the **Extrude** dialog box. However, in some cases you may have a need to move a certain feature face, without editing existing features. For example, you might want to explore a design option, provide needed clearance, or replicate a manufacturing process, such as creating a weld gap. These types of applications can be accomplished using the **Move Face** tool, which removes or adds material to a selected face, without eliminating existing feature parameters. To access the **Move Face** tool and open the **Move Face** dialog box, pick the **Move Face** button on the **Part Features** panel bar, or pick the **Move Face** button on the **Part Features** toolbar. See Figure 6-44.

Figure 6-44.
The **Move Face** dialog box, **Direction Distance** option.



You can choose to move a face using the **Direction Distance** option or the **Planar Move** option. By default, the **Distance and Direction** option is active and allows you to move a face or group of faces a specified distance, along a linear edge. To move a face using this technique, first pick the **Select Faces** button if it is not already selected. Then, pick the faces that you want to move. Once you select one or more feature faces to move, pick the **Select Edge** button located in the **Distance & Direction** area, and pick the desired edge. The edge defines where the face or faces are moved. For example, if

you select a horizontal edge, the face moves along a horizontal plane. When you select an edge, a preview arrow displays the direction and intended effect of the move. See Figure 6-45. If the direction of the move is not correct, pick the **Flip Direction** button to move the face in the opposite direction. Finally, specify how far you want the face to move by entering or selecting a value in the **Distance** edit box. When complete, pick the **Apply** button to move the selected faces and continue using the **Move Face** tool, or pick the **OK** button to move the selected faces and exit the tool.

You can also move a face using the **Planar Move** method, by picking the **Planar Move** button in the **Move Face** dialog box. See Figure 6-46. The **Planar Move** option allows you to move one or more faces from one point to another on a specified plane. To use the **Planar Move** technique, pick the **Select Faces** button if it is not already selected. Then, pick the faces that you want to move. Once you select one or more faces to move, pick the **Select Plane** button located in the **Direction & Distance** area, and pick the desired plane. The plane defines where the face or faces are moved. Typically, a plane that is not parallel to the specified faces is selected. The next step is to pick the **Select Points** button to define the move distance by specifying points. The face moves from the first point you select to the second point you choose. You can select any available model points, such as endpoints, or work points. You can also define the points using precise input techniques. When a face or faces, a plane, and points are selected, pick the **Apply** button to move the selected faces and continue using the **Move Face** tool, or pick the **OK** button to move the selected faces and exit the tool. See Figure 6-47.



Complete the Exercise on the Student CD.

Figure 6-45.
Moving a face using the **Direction Distance** option.

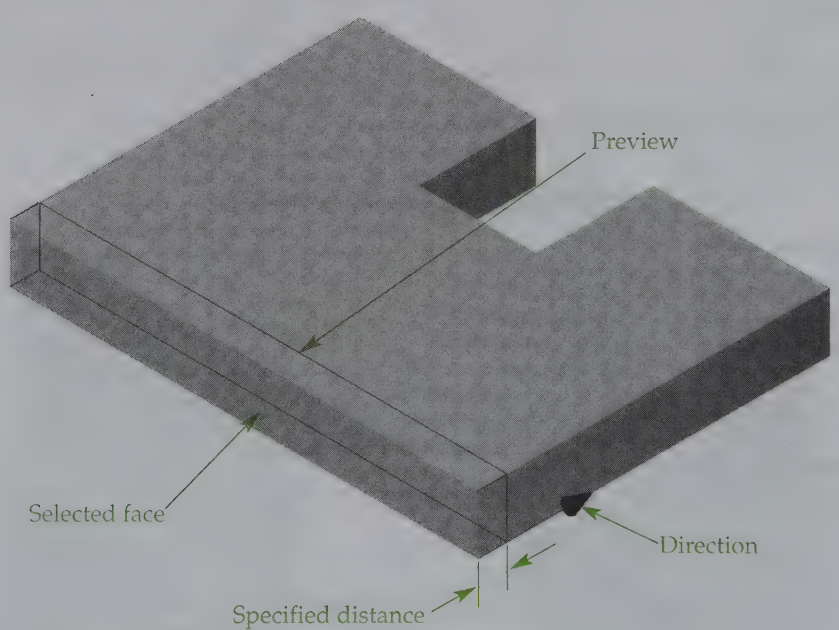


Figure 6-46.
The **Move Face** dialog box, **Planar Move** option.

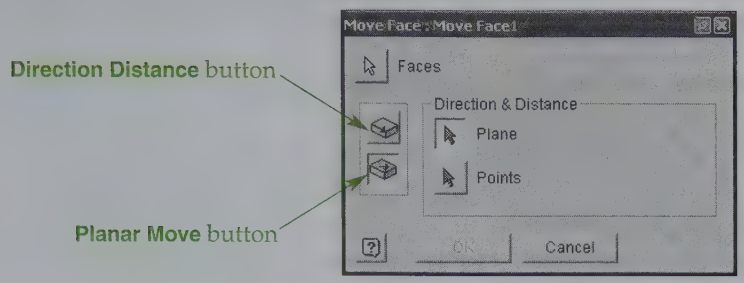
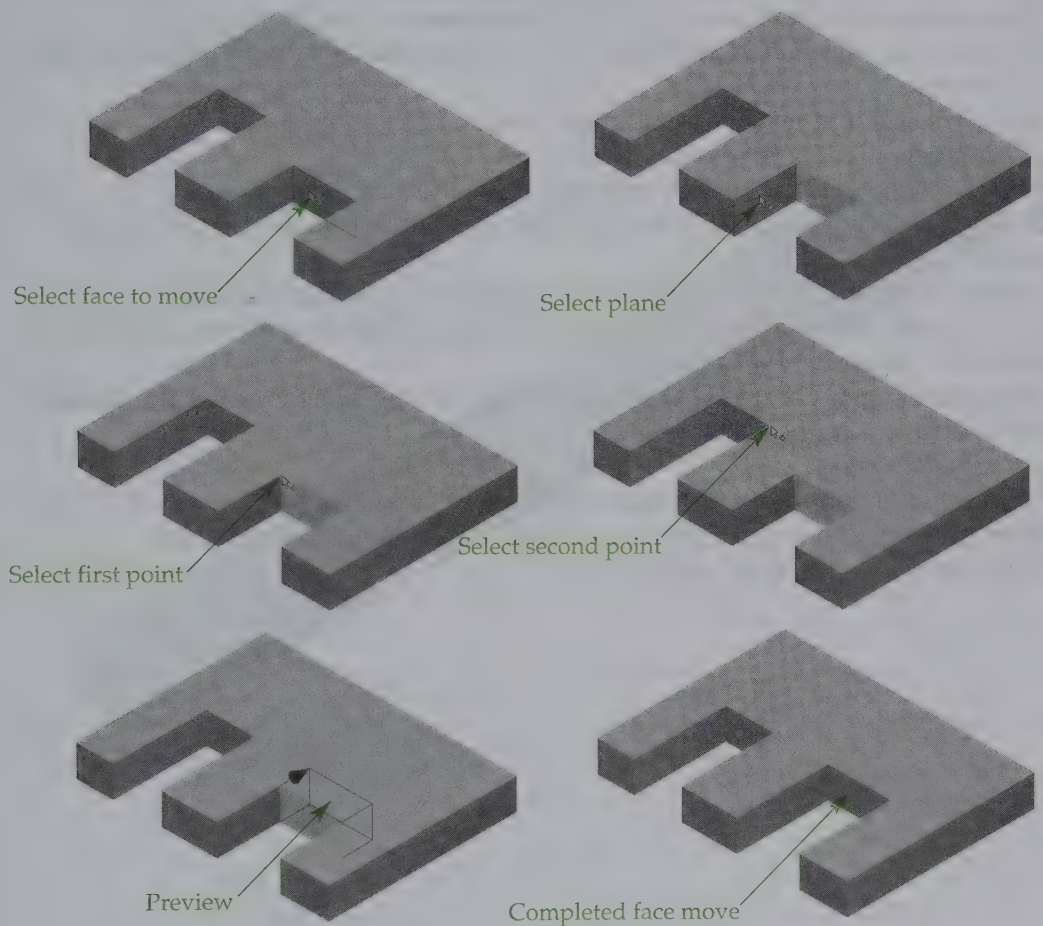
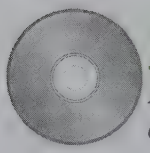


Figure 6-47.
Moving a face using the **Planar Move** option.





Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

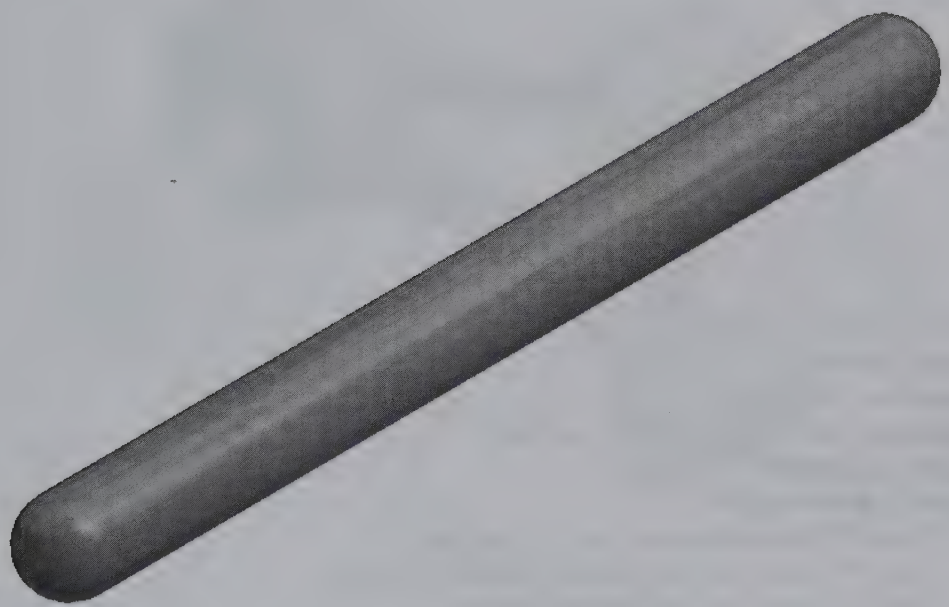
1. What must be present before you can add a *placed feature*?
2. When can placed features be created?
3. Name at least three types of placed features.
4. Define *fillet* and briefly discuss their use.
5. Define *round* and briefly discuss their use.
6. What is a *constant round*?
7. What is a *variable fillet*?
8. Give the function of the **Smooth** radius transition check box.
9. What are setbacks?
10. Explain the effect that happens when a setback is greater than the fillet or round radius, and when a setback is less than or equal to the fillet or round radius.
11. What is selected when creating a face fillet or round?
12. Define a *full radius* and identify the fillet tool type specifically designed for add fully radii.
13. Define *chamfer* and briefly discuss their use.
14. What is a *shell*, and briefly explain their use.
15. Describe a *face draft* and briefly explain where they can be applied.
16. Define *pull direction* and tell what is shown to identify which way the part is pulled.
17. Define *threads* and give an example of internal and external threads.
18. How is pitch determined for ANSI Unified Screw Threads, and for ANSI Metric M Profile threads?
19. What does it mean to say that thread representations, created using the **Thread** tool, are fully parametric?
20. Describe *thickening*.
21. Define *offsetting* in the part feature environment.
22. Name the tool that adds or removes material to a selected face without eliminating existing feature parameters.

Problems

Instructions: Open the following part files from previous chapters, or create new parts as needed, and follow the specific instructions provided for each problem.

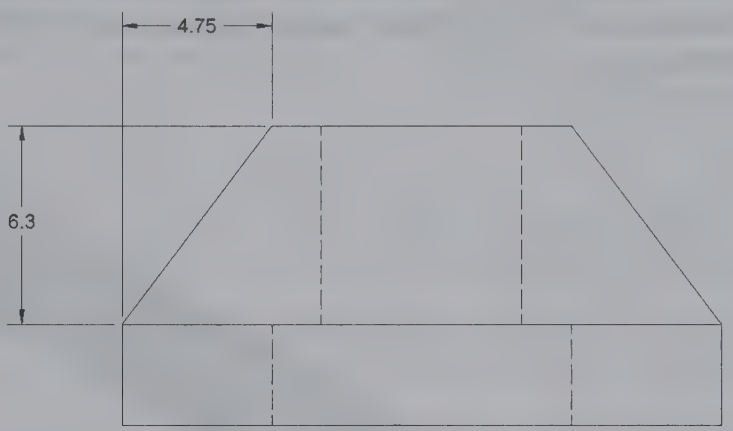
1. **Name:** C-Clamp Pin
   Units: Metric
   Template: PART-mm.ipt
   Material: Steel, High Strength Low Alloy
   Color: As Material
   Specific Instructions:
   - Open P5-1.ipt and save a copy as P6-1.ipt.
   - Close P5-1.ipt without saving and open P6-1.ipt.
   - Place 2.375mm rounds on both ends of the pin as shown.
   - Open the **Properties** dialog box for the model and enter C-CLAMP PIN in the **Title** edit box, PIN in the **Part Number** edit box, and C-CLAMP PIN in the **Description** edit box.

   Save As: P6-1.ipt



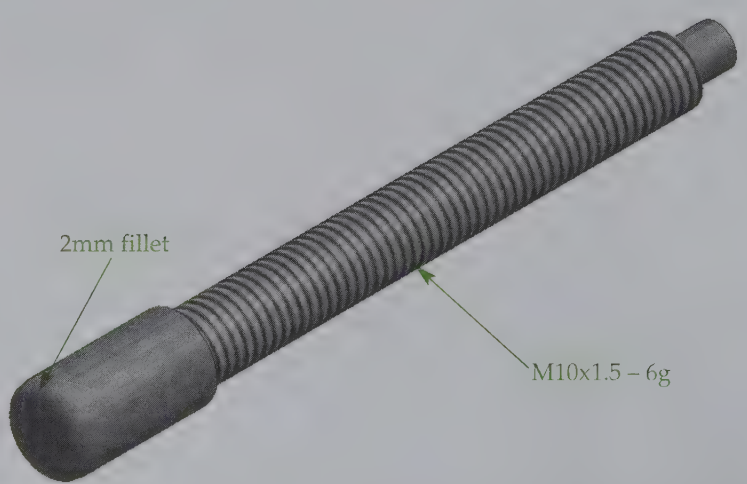
2. **Name:** C-Clamp Swivel
Units: Metric
Template: PART-mm.ipt
Material: Steel, High Strength Low Alloy
Color: As Material
Specific Instructions:
- Open P5-2.ipt and save a copy as P6-2.ipt.
- Close P5-2.ipt without saving and open P6-2.ipt.
- Place a chamfer on the part as shown.
- Open the **Properties** dialog box for the model and enter C-CLAMP SWIVEL in the **Title** edit box, SWIVEL in the **Part Number** edit box, and C-CLAMP SWIVEL in the **Description** edit box.

Save As: P6-2.ipt
Note: Dimensions are for reference only.



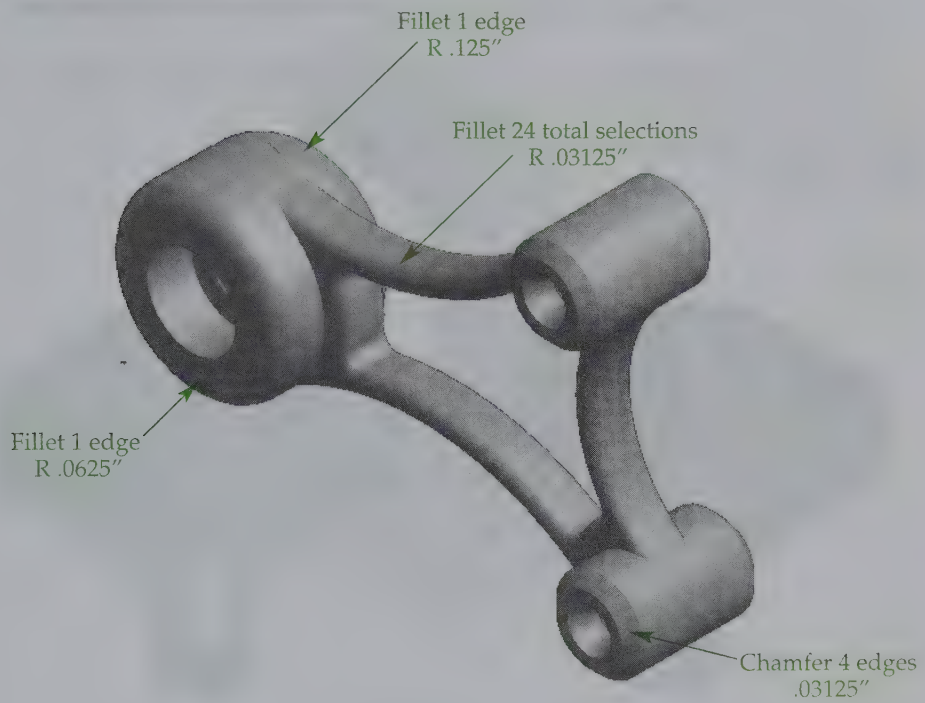
3. **Name:** C-Clamp Screw
Units: Metric
Template: PART-mm.ipt
Material: Steel, High Strength Low Alloy
Color: As Material
Specific Instructions:
- Open P5-3.ipt and save a copy as P6-3.ipt.
- Close P5-3.ipt without saving and open P6-3.ipt.
- Place a 2mm fillet on the edge shown.
- Apply full-length, right-hand threads using the specifications shown.

Save As: P6-3.ipt



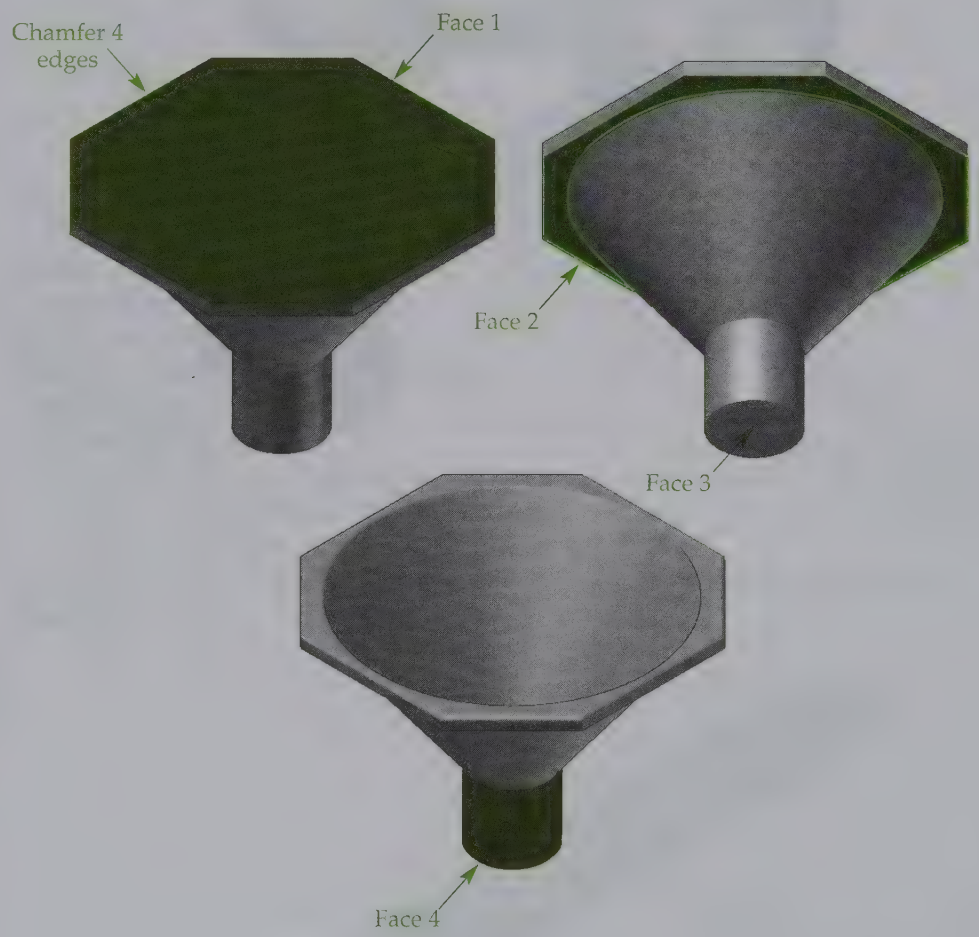
4. **Name:** Support Bracket
   Units: Inch
   Template: PART-in.ipt
   Material: Aluminum-6061
   Color: As Material
   Specific Instructions:
   - Open P5-4.ipt and save a copy as P6-4.ipt.
   - Close P5-4.ipt without saving and open P6-4.ipt.
   - Apply the fillets shown.
   - Apply the chamfers shown.
   - Open the **Properties** dialog box for the model and enter SUPPORT BRACKET in the **Title** edit box, and P6-4 in the **Part Number** edit box.

   Save As: P6-4.ipt



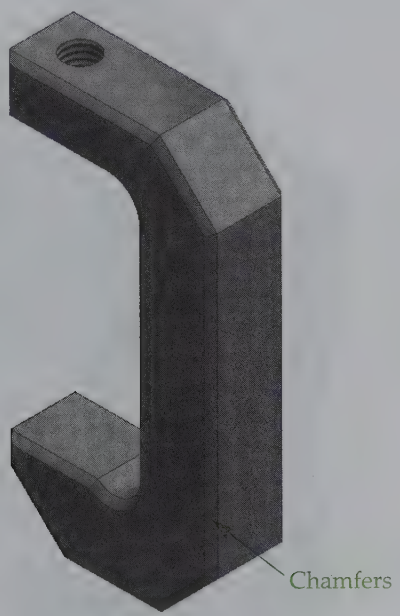
5. **Name:** Funnel
Units: Inch
Template: PART-in.ipt
Material: Acetal Resin, Black
Color: As Material
Specific Instructions:
- Open P5-5.ipt and save a copy as P6-5.ipt.
- Close P5-5.ipt without saving and open P6-5.ipt.
- Place four .75″ chamfers as shown.
- **Shell the part using the following techniques:**
  - Remove Face1 and Face2.
  - Use an **Inside** shell method.
  - Apply a thickness of .0625″.
  - Apply a unique thickness of .125″ to Face3 and Face4.
- Place .03125″ fillets and rounds using the **All Fillets** and **All Rounds** check boxes.
- Open the **Properties** dialog box for the model and enter FUNNEL in the **Title** edit box, and P6-5 in the **Part Number** edit box.

The final part should look like the part shown.
Save As: P6-5.ipt



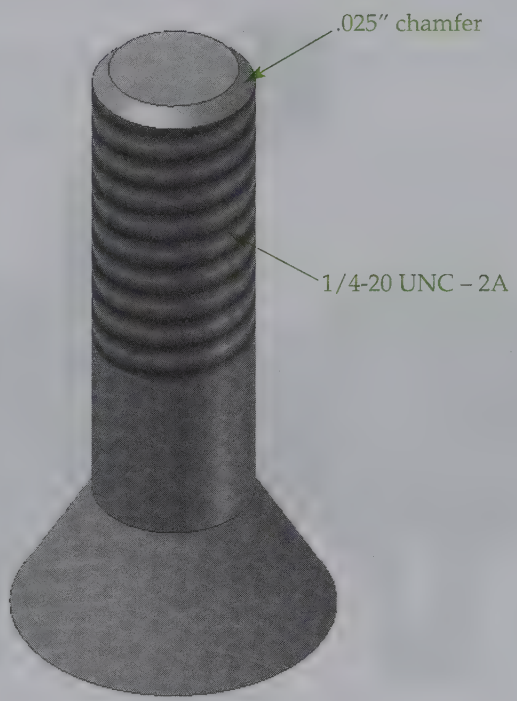
6. **Name:** C-Clamp Body
   Units: Metric
   Template: PART-mm.ipt
   Material: Steel, High Strength Low Alloy
   Color: Red
   Specific Instructions:
   - Open P5-6.ipt and save a copy as P6-6.ipt.
   - Close P5-6.ipt without saving and open P6-6.ipt.
   - Place 1.6mm chamfers on the edges shown, 26 total if the single edge button is selected.
   - Open the **Properties** dialog box for the model and enter C-CLAMP BODY in the **Title** edit box, BODY in the **Part Number** edit box, and C-CLAMP BODY in the **Description** edit box.

   Save As: P6-6.ipt



7. **Name:** Torsion Spring Swivel Bolt
Units: Inch
Template: PART-in.ipt
Material: Default
Color: Default
Specific Instructions:
- Open P5-9.ipt and save a copy as P6-7.ipt.
- Close P5-9.ipt without saving and open P6-7.ipt.
- Thread the bolt stud using the specifications shown. The threads should be right hand, .5" in length, and have an offset of 0.
- Place a .025" chamfer on the edge shown.

The final project should look like the part shown.
Save As: P6-7.ipt



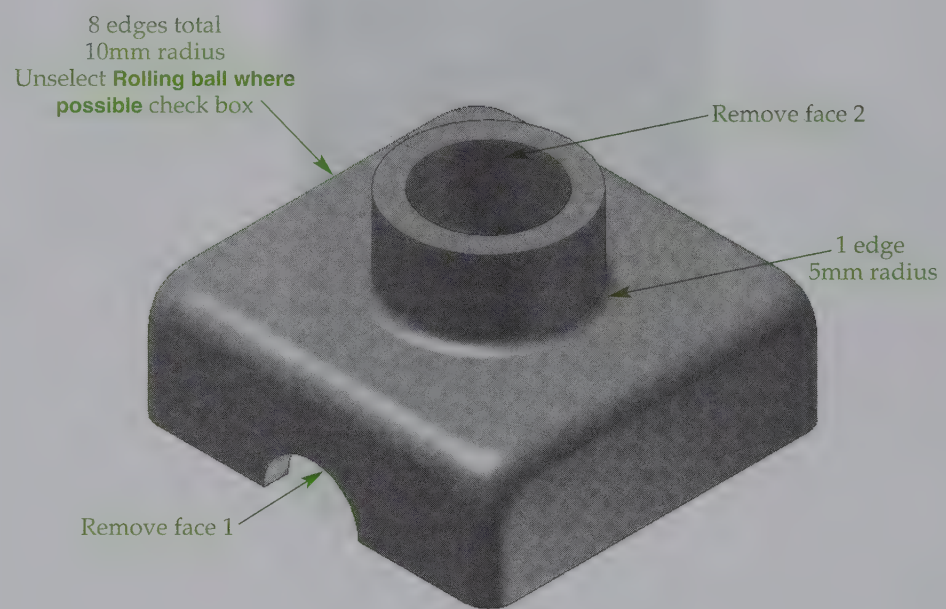
8. **Name:** Bottle
   Units: Metric or Inch
   Template: PART-in.ipt or PART-mm.ipt
   Material: Default or your choice
   Color: Default or your choice
   Specific Instructions:
   - Use the following tools to create a bottle similar to the bottle shown:
     - 2 Extrusions
     - Constant fillets
     - Variable fillets
     - 1 Shell
     - 1 Threaded feature

   Save As: P6-8.ipt



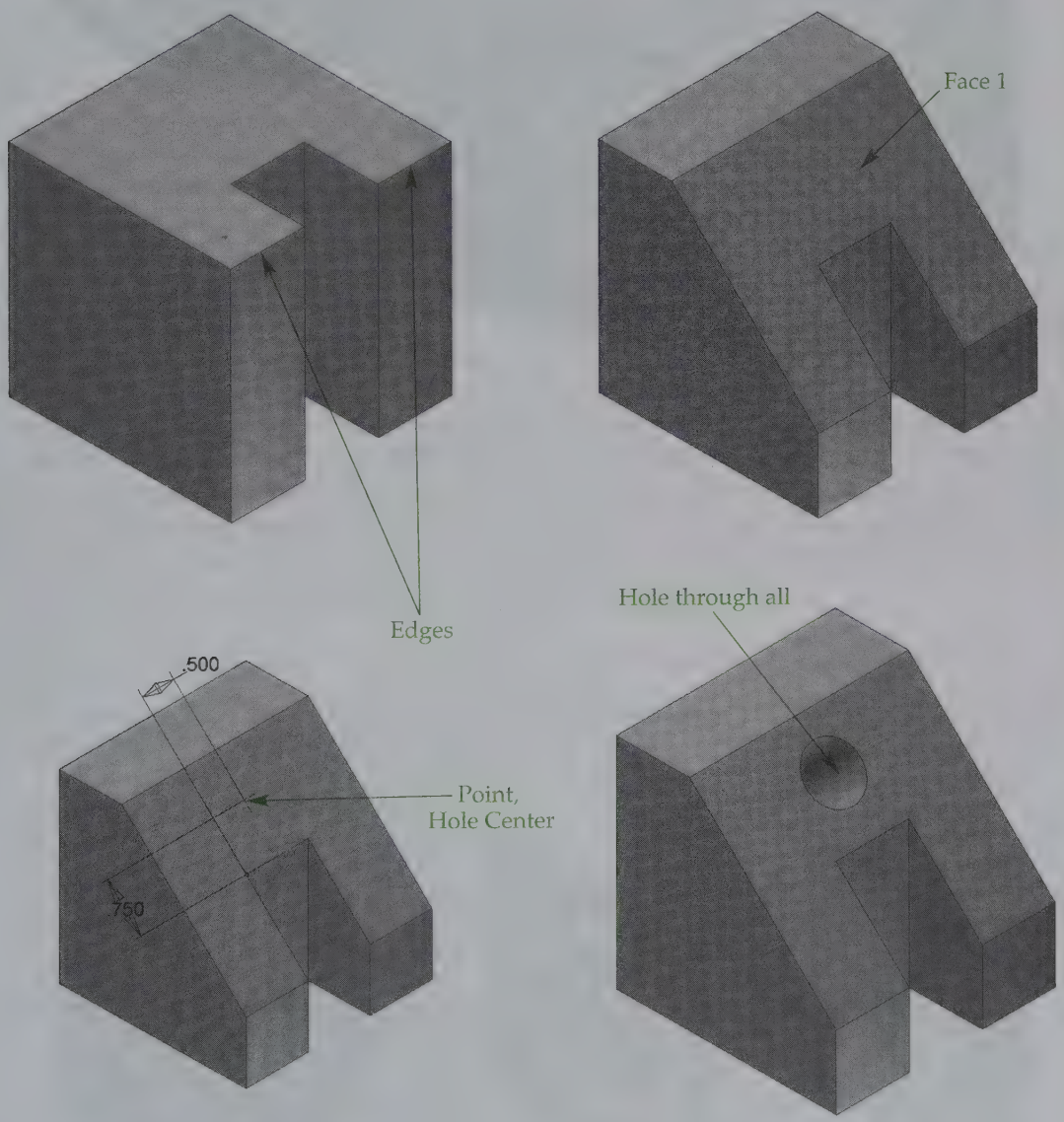
9. **Name:** Housing
Units: Metric
Template: PART-mm.ipt
Material: Cast Steel
Color: As Material
Specific Instructions:
- Begin a new metric part file.
- Open a sketch on the XZ plane.
- Extrude a 150mm X 145mm rectangle 50mm in a positive direction.
- Sketch a ∅70mm circle in the center of the top face of the extrusion.
- Extrude the circle 35mm in a positive direction.
- Fillet the specified edges using the provided information.
- Shell the features using the following information:
  - Remove Faces l and 2
  - Thickness: 10mm
  - Direction: Inside
- Sketch a 20mm radius arc in the center of the face shown.
- Cut-extrude the arc using the **To Next** termination option.

The final project should look like the part shown.
Save As: P6-9.ipt



10. **Name:** Support Block
Units: Inch
Template: PART-in.ipt
Material: Default
Color: Default
Specific Instructions:
- Open P4-7.ipt and save a copy as P6-10.ipt.
- Close P4-7.ipt without saving and open P6-10.ipt.
- Extrude the sketch 3″ in a positive direction as shown.
- Place a 2″ chamfer on the edges shown.
- Open a sketch on Face 1, and sketch the geometry shown.
- Add a .75″ hole through the entire part.
- Open the **Properties** dialog box for the model and enter PROBLEM 6-10 in the **Title** edit box, and P6-10 in the **Part Number** edit box.

The final part should look like the part shown.
Save As: P6-10.ipt



This model has many placed features. These features do not require a sketch, and therefore, save time. See if you can identify the placed features in this model of a machined casting.



CHAPTER 7

Feature Patterns

Learning Objectives

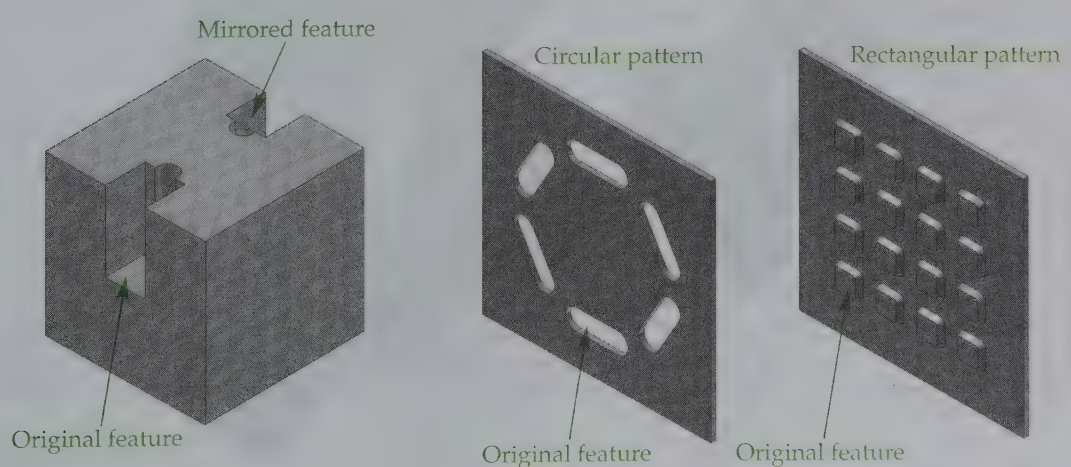
After completing this chapter, you will be able to do the following:

- Mirror features.
- Create rectangular patterns.
- Develop circular patterns.

As you continue developing part models, you may need to create multiple arranged copies, or patterns, of features. A reoccurrence of a feature or features, in a designated configuration, is known as a ***feature pattern***. Inventor provides you with tools that allow you to pattern a feature or group of features by mirroring, patterning in a circular fashion, or patterning in a rectangular fashion. See Figure 7-1.

Pattern features can be created anytime during part model development, as long as suitable features exist. Typically, features are patterned when they are complete, such as patterning an extrusion that contains a chamfer and a hole. You cannot pattern placed features such as fillets, chamfers, shells, face drafts, and threads by themselves. However, you can pattern sketched features and include placed features that have been added to the sketch's features when generating a pattern.

Figure 7-1.
Mirrored features, circular patterns, and rectangular patterns are quick methods of creating symmetrical and multiple copies of features.



Mirroring Features

A *mirrored feature* consists of one or more features that are mirrored symmetrically over a specified plane. Mirroring is used to create a reversed copy of features for symmetrical parts, without having to develop additional sketched, placed, or catalog features. Figure 7-2 shows an example of a part created by mirroring an extrusion, fillets, and a hole. Mirrored features are created using the **Mirror Feature** tool, available by picking the **Mirror Feature** button on the **Part Features** panel bar; picking the **Mirror Feature** button on the **Part Features** toolbar; or pressing [Shift]+[M].

The **Mirror Pattern** dialog box is displayed when you access the **Mirror Feature** tool. See Figure 7-3. You can use the **Mirror Feature** tool to mirror a single feature, or a group of features. To mirror individual features, use the **Mirror individual features** button, which should be selected by default. Then, pick the **Features** button if it is not already active, and select the features you want to mirror. You may need to pick certain features from the browser if you cannot access them from the graphics window. Once you select all the features you want to mirror, pick the **Mirror Plane** button and select the mirror plane. The *mirror plane* acts as the line of symmetry, defining where the specified features are mirrored across. You can choose any available plane, including planar feature faces, work planes, and the default reference planes located in the **Origin** folder of the browser.

You also have the option of selecting all of the features currently in the part by choosing the **Mirror the entire solid** button. Though you can pick all the part features using the **Mirror individual features** option, if you want to mirror the complete part, pick the **Mirror the entire solid** button. See Figure 7-3. Then, pick the **Mirror Plane** button, if it is not already active, and select the mirror plane. Using the **Mirror the entire solid** option automatically selects all of the features found in the part, but not work features. In order to include work features in the mirror operation, pick the **Include Work/Surface Features** button and select the work feature you want to mirror. You also have the option of eliminating the original part from the display by selecting the **Remove Original** check box. Picking the **Remove Original** check box replaces the original features with the mirrored features, but does not actually delete the initial part.

Figure 7-2.
An example of a part created by mirroring an extrusion, fillets, and a hole.

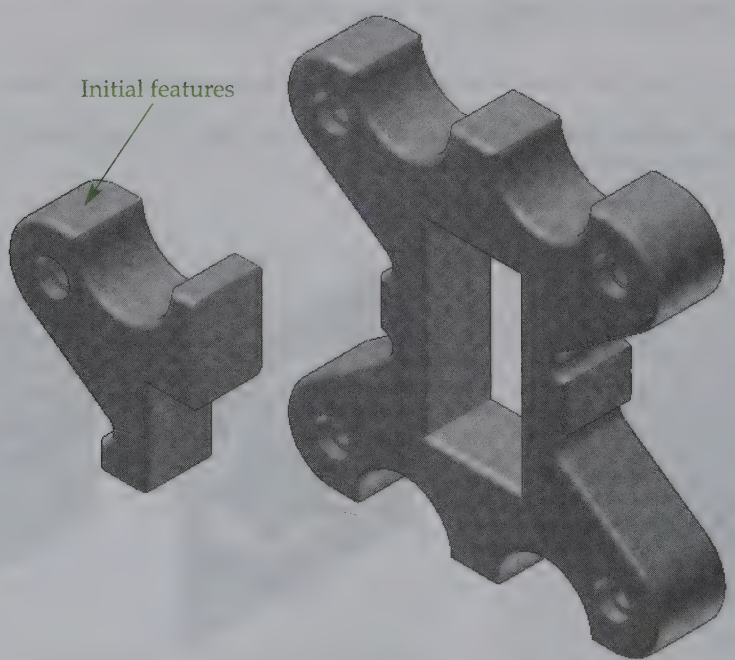
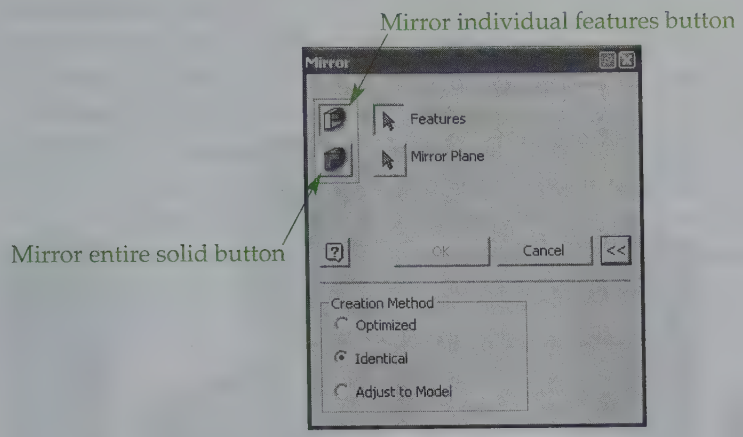


Figure 7-3.
The **Mirror Pattern** dialog box, **Mirror individual features** button.



After you select individual features or the entire model and pick a mirror plane, you may want to change the mirror creation method by picking the **More** button and accessing the **Creation Method** area. Selecting the **Optimized** radio button allows you to create an exact copy of the selected features shape, but reproduces the features as faces, not features. All characteristics of the parent features, including the specified termination option, are reproduced in the mirrored faces.

Choosing the **Identical** radio button allows you to create mirrored features that are exactly the same as the parent features. The **Identical** creation method functions the same as the **Optimized** method, but reproduces actual features, instead of just faces. All characteristics of the parent feature, including the specified termination option, are reproduced in the mirrored feature. Select the **Identical** radio button when the features you are mirroring terminate on a work plane or the same planar face.

Picking the **Adjust to Model** radio button allows you to create mirrored features that are based on the parent features, but change size to adjust for variations in the model, especially when the parent feature terminates on a feature face. For standard mirroring operations you do not need to select the **Adjust to Model** radio button.

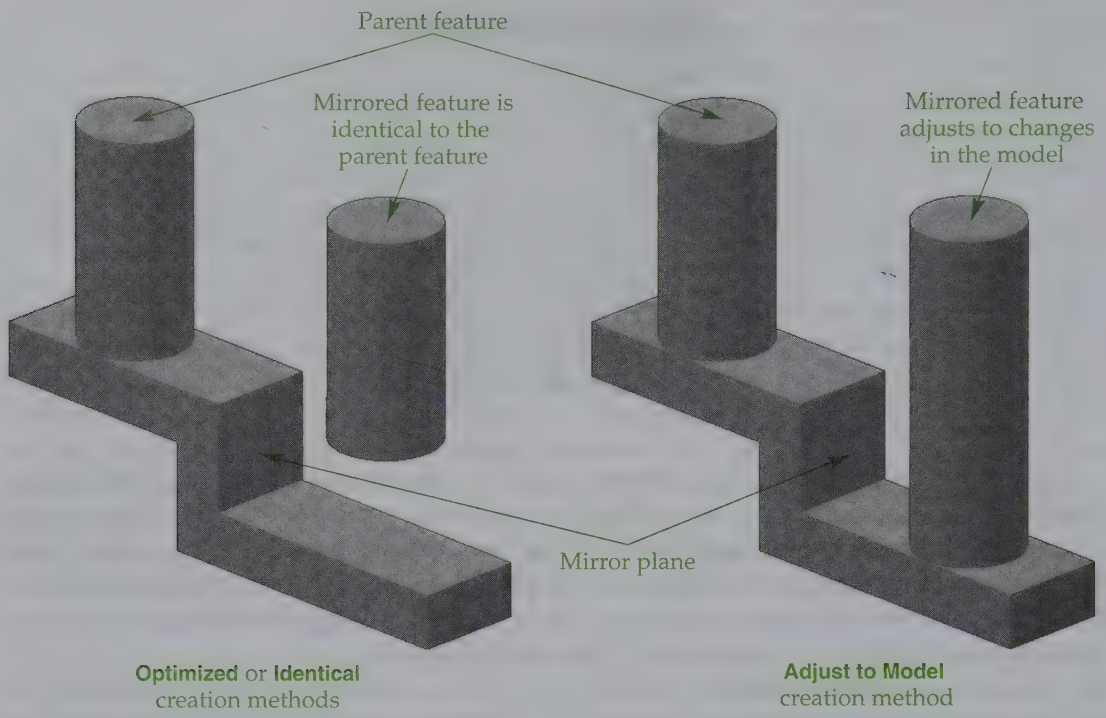
Figure 7-4 shows the difference between using an **Optimized** or **Identical** creation method and the **Adjust to Model** method.

PROFESSIONAL TIP



The **Optimized** creation method generates mirrored features the quickest. As a result, you may want to select this radio button for mirroring complex geometry, unless the features must change termination specifications as they are reproduced or you want to recreate features. The **Identical** creation method generates mirrored features quickly. As a result, you should select this radio button for mirroring complex geometry, unless the features must change termination specifications as they are reproduced. The **Adjust to Model** creation method takes longer than the **Optimized** and **Identical** creation methods to generate mirrored features. Select the **Adjust to Model** radio button only when mirrored features must modify to fit changes in model geometry.

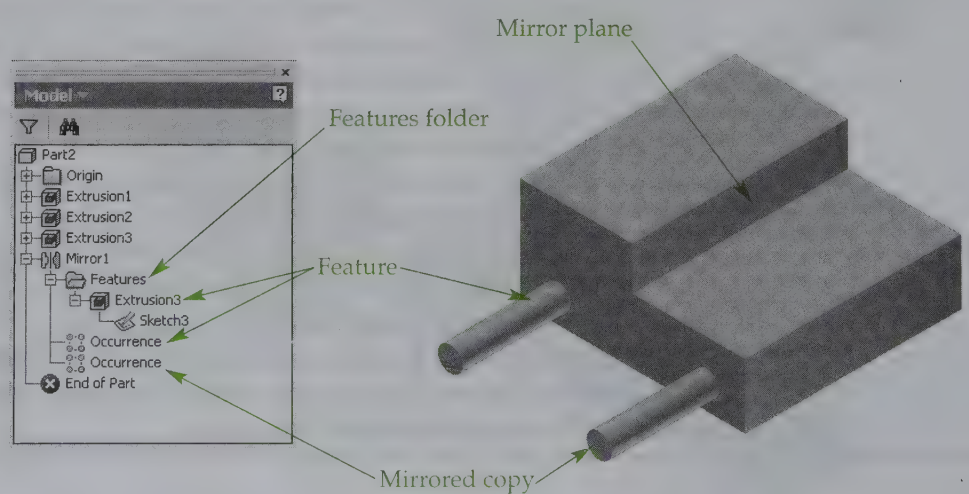
Figure 7-4.
Mirroring a feature using the **Optimized** or **Identical** creation methods and the **Adjust to Model** creation method.



Once you select the features you want to mirror, pick the mirror plane, and choose the desired creation method, pick the **OK** button to mirror the features.

Figure 7-5 shows an example of the information displayed in the browser bar when you mirror a feature. The mirror feature is identified by the mirror icon and is named Mirror1, by default, for the first mirror feature. As you create additional mirrored features, the numbers next to Mirror increase. The mirrored feature contains the following information used to generate the mirrored geometry:

Figure 7-5.
Compare the items in the browser to the related items in a model containing a mirrored feature.



- **Features.** This folder contains the parent features and sketches that have been mirrored. The feature mirrored in Figure 7-5 contains an extrusion, and the extrusion sketch.
- **Occurrences.** Every time you pattern a feature, including mirrored features, the occurrences are listed under the mirror feature. *Occurrences* identify how many features are present as a result of the pattern operation. For mirrored features, only two occurrences are present, the parent feature and the mirrored feature. When you move your cursor over the occurrence, the feature is highlighted in the graphics window. If you want to suppress, or "turn off," the mirror feature, right-click the correct occurrence and pick the **Suppress** menu option.

NOTE



You cannot suppress the parent feature occurrence, but you can suppress the actual parent feature by right-clicking on the feature and selecting the **Suppress Feature** pop-up menu option.

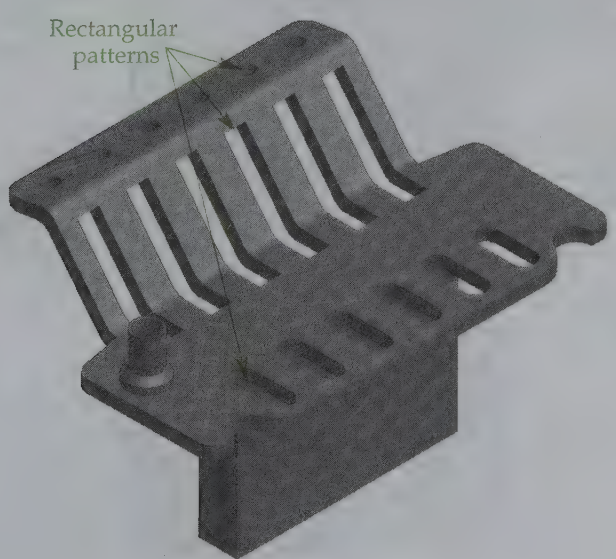


Exercise 7-1 Complete the Exercise on the Student CD.

Creating Rectangular Feature Patterns

A ***rectangular feature pattern*** is one or more features arranged in a rectangular configuration. A rectangular pattern consists of a number of features, or group of features, positioned a specified distance apart, in rows and columns. Rectangular patterning is used to arrange multiple copies of features, without having to develop additional sketched, placed, or catalog features. Figure 7-6 shows some examples of using rectangular patterning. Rectangular patterns are created using the **Rectangular Pattern**

Figure 7-6.
Examples of rectangular patterns.



dialog box. See Figure 7-7. Access the **Rectangular Pattern** tool by picking the **Rectangular Pattern** button on the **Part Features** panel bar; picking the **Rectangular Pattern** button on the **Part Features** toolbar; or pressing [Shift]+[R].

To pattern individual features, use the **Pattern individual features** button, which should be selected by default. Then, pick the **Features** button if it is not already active, and select the features you want to pattern. You may need to pick certain features from the browser if you cannot access them from the graphics window.

You also have the option of selecting all of the features currently in the part by choosing the **Pattern the entire solid** button. Though you can pick all the part features using the **Pattern individual features** option, if you want to pattern the complete part, select the **Pattern the entire solid** button as shown in Figure 7-7. Using the **Pattern the entire solid** option automatically selects all of the features found in the part, except for work and surface features. In order to include work features in the pattern operation, pick the **Include Work/Surface Features** button and select the work feature you want to pattern.

Once you select all the features you want to pattern, pick the **Path** button, located in the **Direction 1** area, and select the path direction of the pattern row or column. You can select any straight or curved feature edge, axis, or work axis available in the model, including the default reference axes located in the browser bar **Origin** folder. When you choose a path direction, an arrow shows you the specified pattern path direction, and a preview displays the pattern operation. See Figure 7-8. The default pattern path direction can be reversed by picking the **Flip** button. Patterns occur in a direction on one side or the other of the selected features depending on the flip direction. An alternative is to pick the **Midplane** check box to spread the total number of features, or count, on each side of the selected features. You may still need to pick the **Flip** button if you enter an even number of total features. See Figure 7-9.

When you have defined the path direction of the first pattern group, specify the total number of features you want to create in the pattern by entering or selecting a count value in the **Count** edit box. Then define the distance between the copies using one of the following techniques:

Figure 7-7.
The **Rectangular Pattern** dialog box, **Pattern individual features** option.

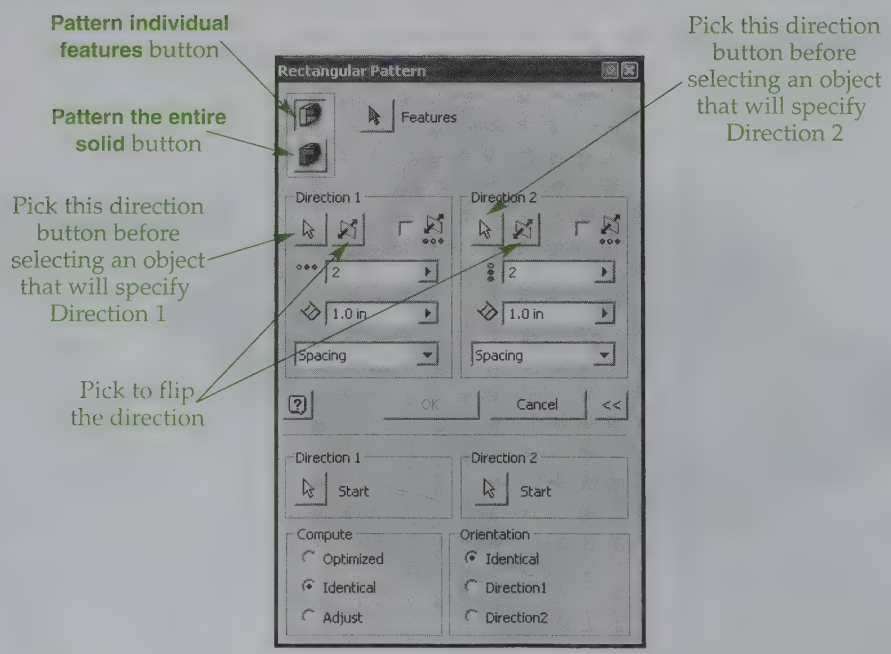


Figure 7-8.
The pattern direction arrow and pattern preview.

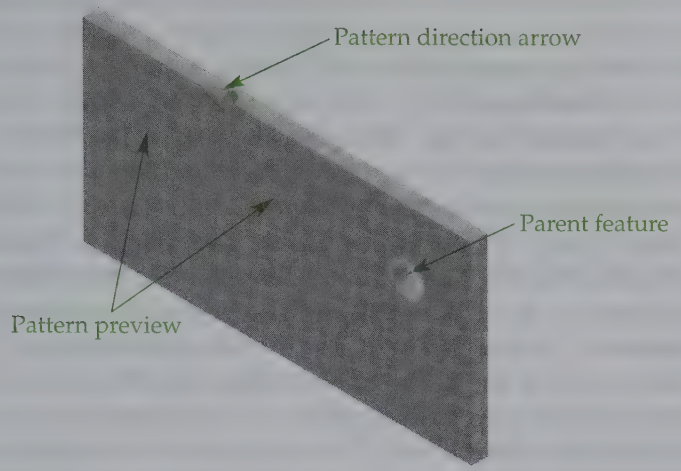
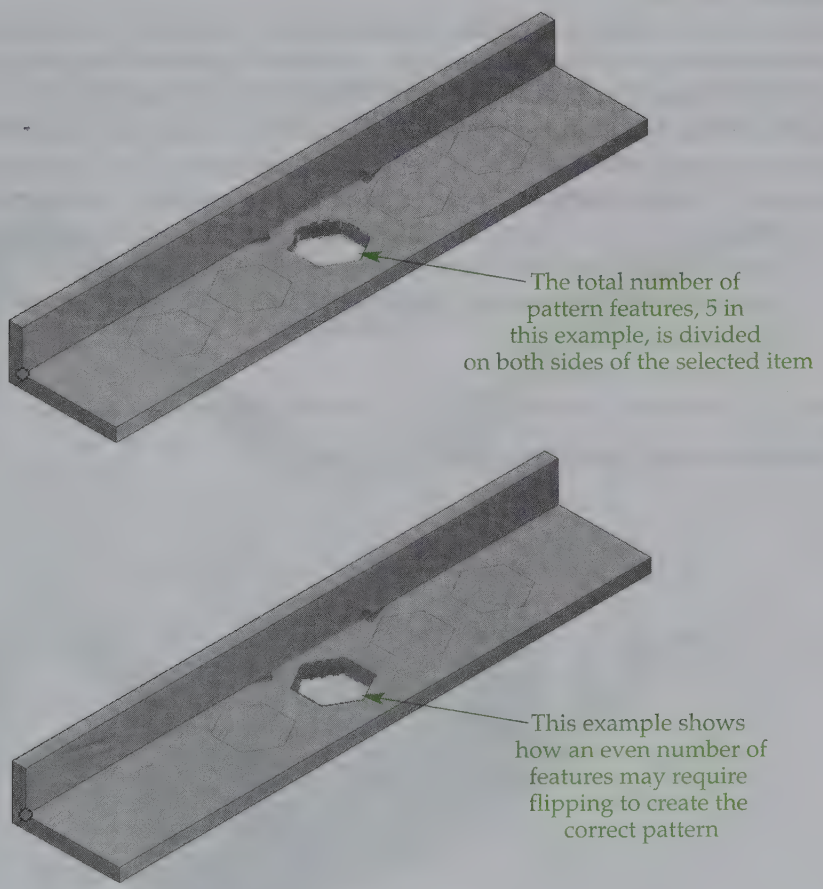


Figure 7-9.
Using the **Midplane** direction option when creating a rectangular pattern.



- Pick the **Spacing** option from the drop-down list, and enter or select the spacing using the **Length** edit box. ***Spacing*** is measured by the width of the selected features and the distance between the copies. For example, if you want to pattern an extrusion that is 1" (24.5 mm) wide and you want a 1/2" (12.25 mm) space between pattern copies, you must specify a 1.5" (36.75 mm) spacing.

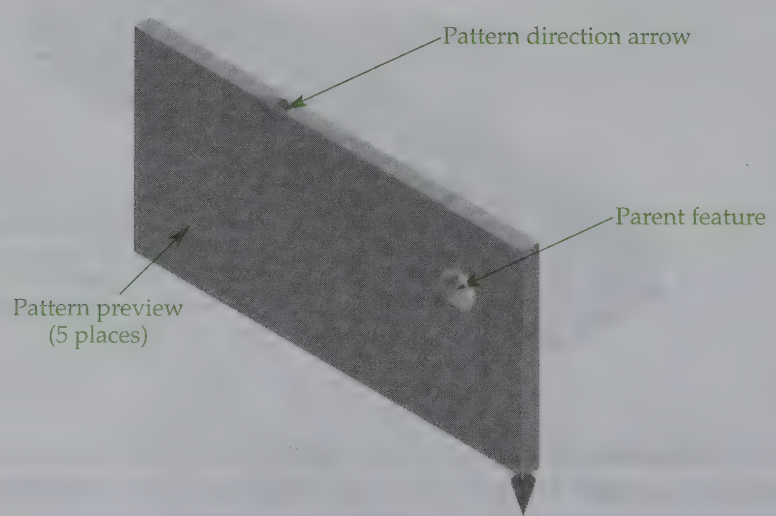
- Select the **Distance** option from the drop-down list and enter or select a distance using the **Length** edit box. Distance is measured from the point on the first pattern occurrence to the corresponding point on the farthest pattern occurrence. For example, the length between the center of the first cylinder in a pattern to the center of the last cylinder in a pattern defines the distance.
- Pick the **Curve Length** option from the drop-down list. When you choose the **Curve Length** option, the entire length of the selected path, or curve, is calculated and defined in the **Length** edit box.

Once you fully define the first pattern row or column using the options in the **Direction 1** area, repeat the steps previously described for defining the settings in the **Direction 2** area. Pick the **Path** button, located in the **Direction 2** area, and select the desired path direction. If the first direction, specified in the **Direction 1** area, defined the rows, **Direction 2** defines the columns. If the first direction, specified using the **Direction 1** area, defined the columns, **Direction 2** defines the rows. Again, you can select any straight or curved feature edge, axis, or work axis available in the model, including the default reference axes located in the browser bar **Origin** folder. You can choose the same edge or axis you chose for **Direction 1**, or a parallel edge or axis. However, for most applications, you should select an edge or axis that is perpendicular, or at least nonparallel to the first direction. When you choose the second direction, an arrow shows you the specified pattern path, and a preview displays the pattern operation. See Figure 7-10.

After you define the direction of the second pattern path, specify how many copies of the feature or series of features you want to create by entering or selecting a count value using the **Count** edit box. Then define the distance between the copies using one of the techniques previously described. If you are generating a more complex rectangular pattern, you may need to adjust some of the following options in the **More** area of the **Rectangular Pattern** dialog box by picking the **More** button:

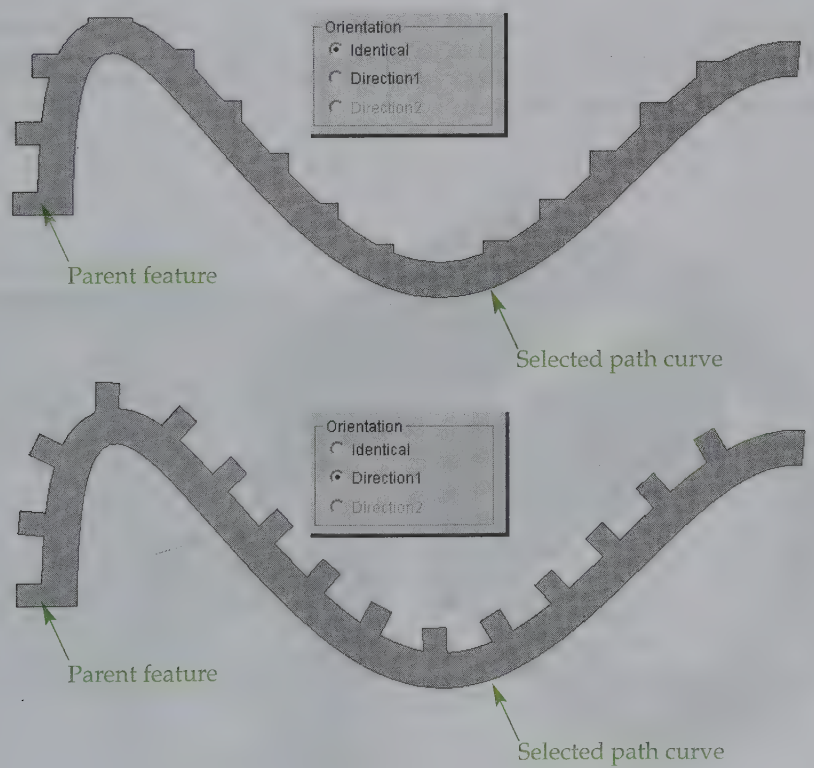
- **Direction 1.** By default, the **Direction 1** pattern starts at the beginning of the selected curve path in reference to the specified direction. If you want to change where the pattern begins, pick the **Direction 1** button and pick a new point on the selected path curve.

Figure 7-10.
The pattern direction arrow and pattern preview.



- **Direction 2.** By default, the **Direction 2** pattern starts at the beginning of the selected curve path, in reference to the specified direction. If you want to change where the pattern begins, pick the **Direction 2** button and pick a new point on the selected path curve.
- **Compute.** This area contains the **Optimized**, **Identical**, and **Adjust** radio buttons described in the **Mirroring Features** section of this chapter. Again, when you select the **Optimized** or **Identical** radio button, all occurrences of the pattern are copied and terminate exactly the same as the parent feature. Select one of these options when the features you are patterning terminate on a work plane or the same planar face. When you pick the **Adjust** radio button, each occurrence of the pattern generates and terminates independently of the parent features, adjusting to model variations. For most typical rectangular pattern operations you do not need to select the **Adjust** radio button.
- **Orientation.** This area contains options for defining the occurrences of rectangular pattern features over complex, often tangent, curves. Pick the **Identical** radio button if you do not want to compensate for a complex curve, and create each pattern occurrence identical to the parent feature. If you do want the rectangular pattern to adapt to a complex curve by rotating each pattern occurrence in reference to a tangent curve, pick the **Direction1** radio button for the **Direction 1** path, and the **Direction2** radio button for the **Direction 2** path. See Figure 7-11.

Figure 7-11.
A rectangular pattern of features with the **Identical** radio button selected and the **Direction 1** or **Direction 2** radio button selected (only one direction is shown).



Now, pick the **OK** button to generate the rectangular pattern.

The browser options and applications previously described for a mirrored feature also apply to rectangular patterns. Rectangular patterns are identified by the rectangular pattern icon and the default Rectangular Pattern 1 name. In addition, the **Features** folder and each of the **Occurrences** are listed. See Figure 7-12. Like a mirrored feature, the occurrences show how many features are present as a result of the pattern operation, including the parent feature. Often you may want to "turn off" certain rectangular pattern occurrences by right-clicking on the desired occurrence and picking the **Suppress** menu option. See Figure 7-13.

Figure 7-12.
An example of the information displayed in the browser when a rectangular pattern is created.

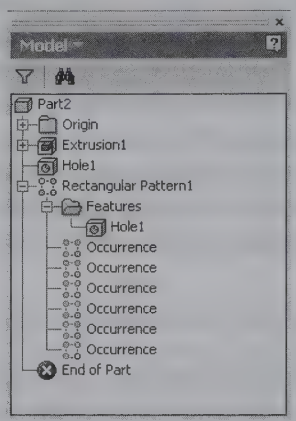
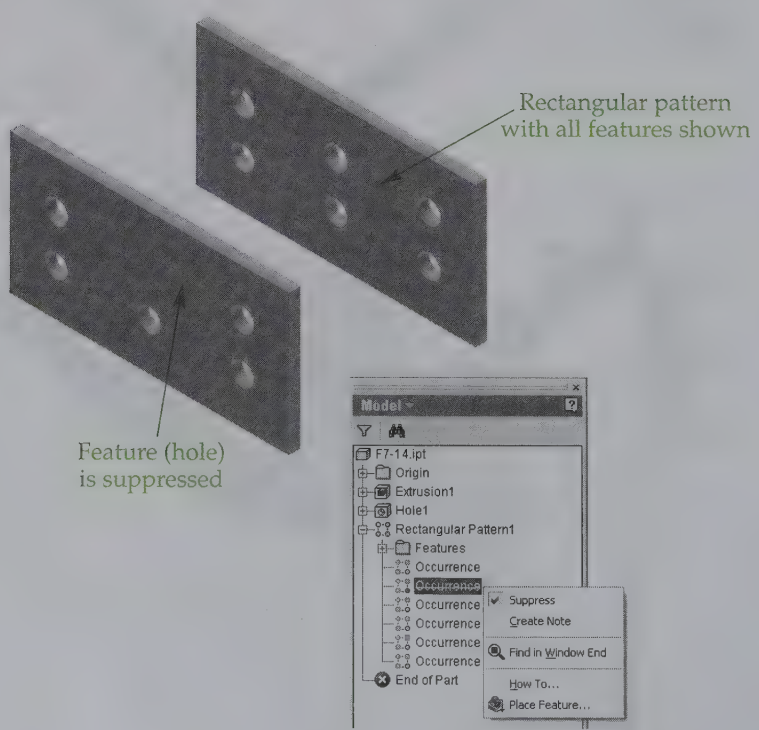


Figure 7-13.
Notice the effects of suppressing a feature.



NOTE

You cannot suppress the parent feature occurrence, but you can suppress the actual parent feature by right-clicking on the feature and selecting the **Suppress Feature** pop-up menu option.

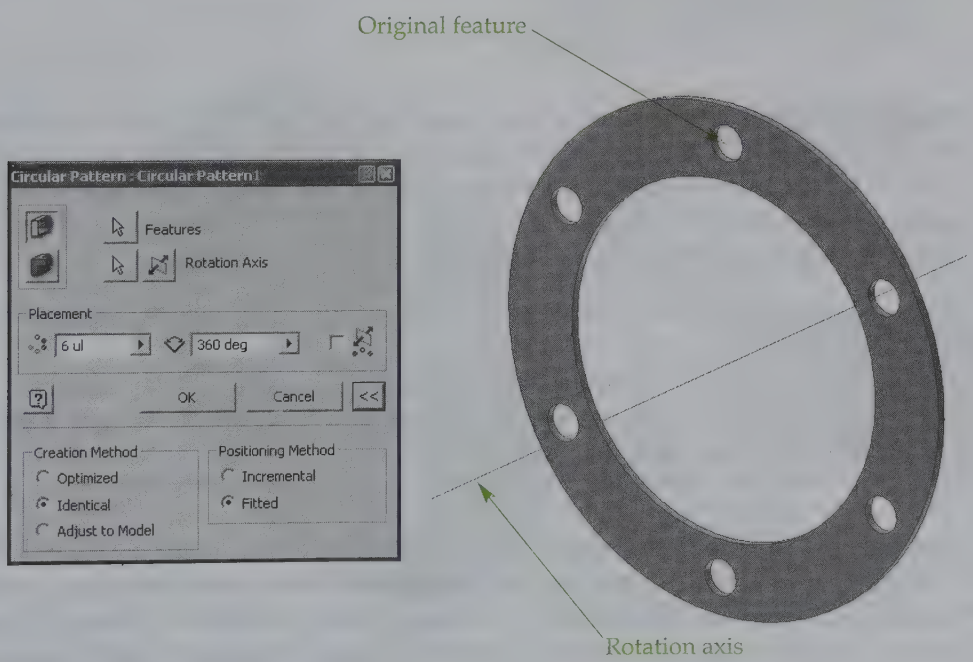
Exercise 7-2

Complete the Exercise on the Student CD.

Creating Circular Feature Patterns

A *circular feature pattern* is created when one or more features are arranged in a circular configuration. A circular pattern consists of a feature, or series of features, positioned a specified distance apart around an axis. Circular patterning is used to arrange multiple copies of features without having to develop additional sketched, placed, or catalog features. Figure 7-14 shows an example of a part created by circular patterning cut extrusions. Circular patterns are created using the **Circular Pattern** tool. To access the **Circular Pattern** tool and open the **Circular Pattern** dialog box, pick the **Circular Pattern** button on the **Part Features** panel bar; pick the **Circular Pattern** button on the **Part Features** toolbar; or press [Shift]+[O].

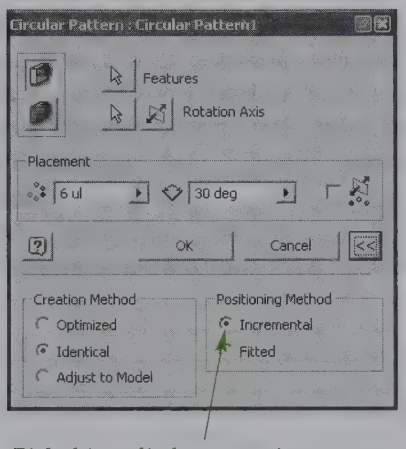
Figure 7-14.
This **Circular Pattern** dialog box was used to copy the original hole about the rotation axis.

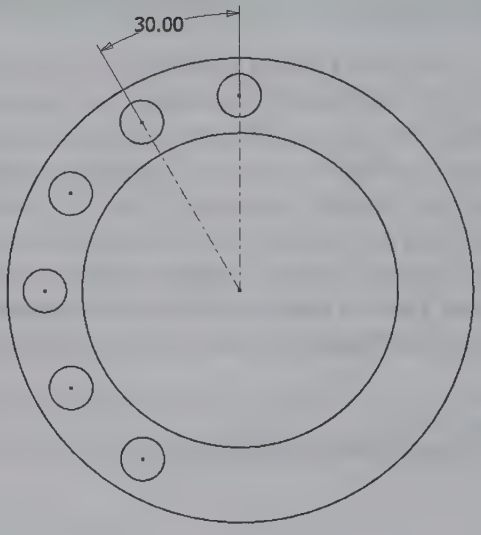


Before you select features to pattern you may first want to change the circular pattern creation and positioning methods by picking the **More** button and accessing the **Creation Method** and **Positioning Method** areas. The **Creation Method** area contains the **Optimized**, **Identical**, and **Adjust to Model** radio buttons previously described in the **Mirroring Features** and **Creating Rectangular Patterns** sections of this chapter. The **Positioning Method** area allows you to specify how the pattern features are placed and contains the following radio buttons:

- **Incremental.** This radio button allows you to specify the distance between features by defining the angle each pattern feature rotates around the selected axis. The total amount of rotation is defined by the number of occurrences and the angle between occurrences. Figure 7-15 shows an example of patterning a feature using a count of 6 and a 30° incremental angle.

Figure 7-15.
Patterning a feature using the **Incremental** option.





- **Fitted.** This radio button allows you to specify the distance between features, by defining the total amount of rotation. The angle between each occurrence is defined by the number of occurrences and the entire revolution angle of all the pattern features. A 360° angle is a full rotation. Figure 7-16 shows an example of patterning a feature using a count of 6 and a 90° fitted angle.

PROFESSIONAL TIP



If you use the **Fitted** option to create a circular pattern, the spacing of pattern feature occurrences changes if you modify the design.

You can use the **Circular Pattern** tool to pattern a single feature, or a group of features. To pattern individual features, use the **Pattern individual features** button, which should be selected by default. See Figure 7-17. Then, pick the **Features** button if it is not already active, and select the features you want to pattern. You may need to pick certain features from the browser if you cannot access them from the graphics window.

You also have the option of selecting all of the features currently in the part by choosing the **Pattern the entire solid** button. Though you can pick all the part features using the **Pattern individual features** option, if you want to pattern the complete

Figure 7-16.
Patterning a feature using the **Fitted** option.

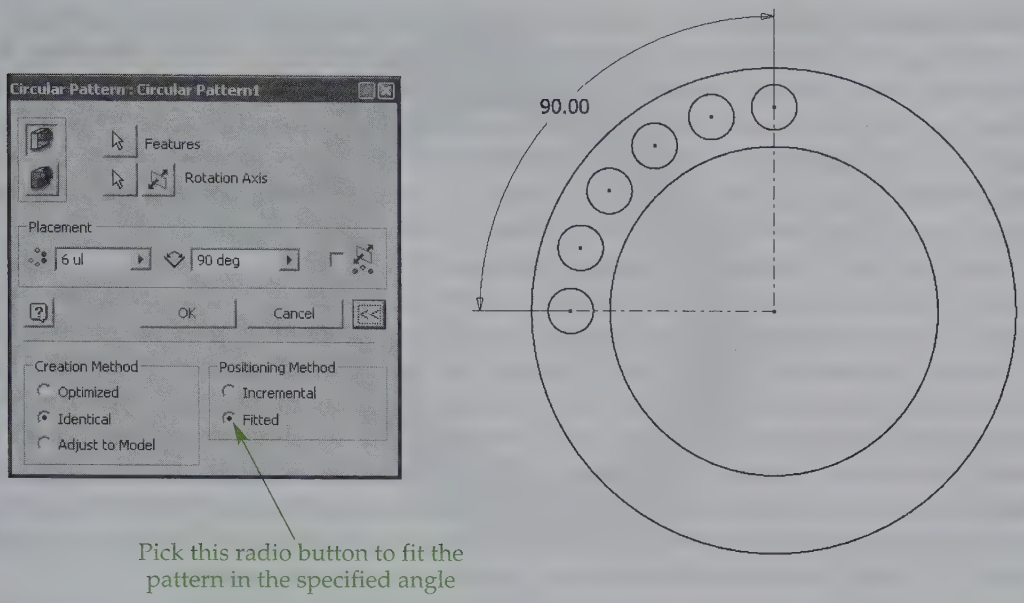
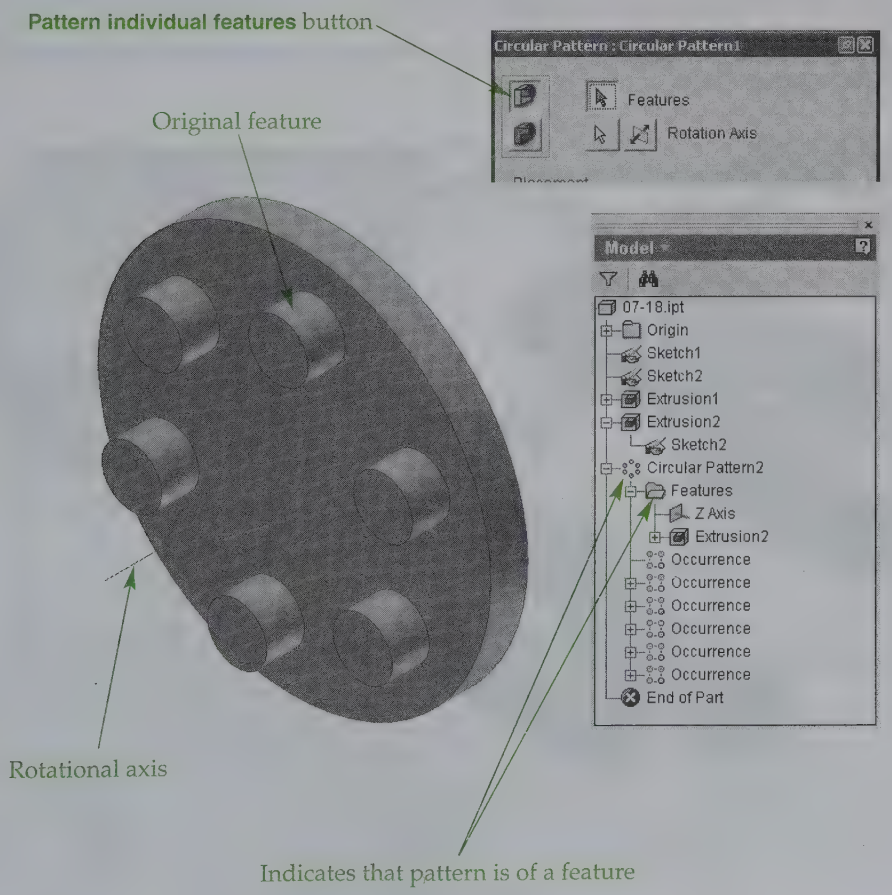


Figure 7-17.
When creating a circular pattern of features, pick the **Pattern individual features** button, create the pattern, and notice the results in the browser.



part, select the **Pattern the entire solid** button as shown in Figure 7-18. Using the **Pattern the entire solid** option automatically selects all of the features found in the part, but not work features. In order to include work features in the pattern operation, pick the **Include Work Features** button and select the work feature you want to pattern.

When all the features you want to pattern are selected, pick the **Rotation Axis** button and select the axis of rotation. The axis of rotation is the center or pivot point that selected features orbit around. You can select any feature edge, axis, or work axis available in the model, including the default reference axes located in the browser **Origin** folder. When you choose a rotation axis, an arrow shows you the specified pattern path, and a preview displays the pattern operation. See Figure 7-19.

Once you select features and define the axis of rotation, specify the total number of features you want to create in the pattern by entering or selecting a value using the **Count** edit box. Then define either the incremental or fitted angle, depending on the radio button you choose, by entering or selecting an angle value using the **Angle** edit box. If the total rotation of all occurrences is not 360° and you want the circular pattern to revolve around the opposite direction, pick the **Flip** button. Patterns occur in a direction on one side or the other of the selected features, depending on the flip direction. An alternative is to pick the **Midplane** check box to spread the total number of features each side of the selected items. You may still need to pick the **Flip** button if you enter an even number of total features. See Figure 7-20. Finally, select the **OK** button to generate the circular pattern.

The browser bar options and applications previously described for a rectangular feature also apply to circular patterns. Circular patterns are identified by the circular pattern icon and the default **Circular Pattern1** name. The **Features** folder and each of the **Occurrences** are also listed. Refer back to Figures 7-16 and 7-17. Like mirrored features and rectangular patterns, circular pattern occurrences define how many features are

Figure 7-18.
When creating a circular pattern of parts, pick the **Pattern the entire solid** button, create the pattern, and notice the results in the browser.

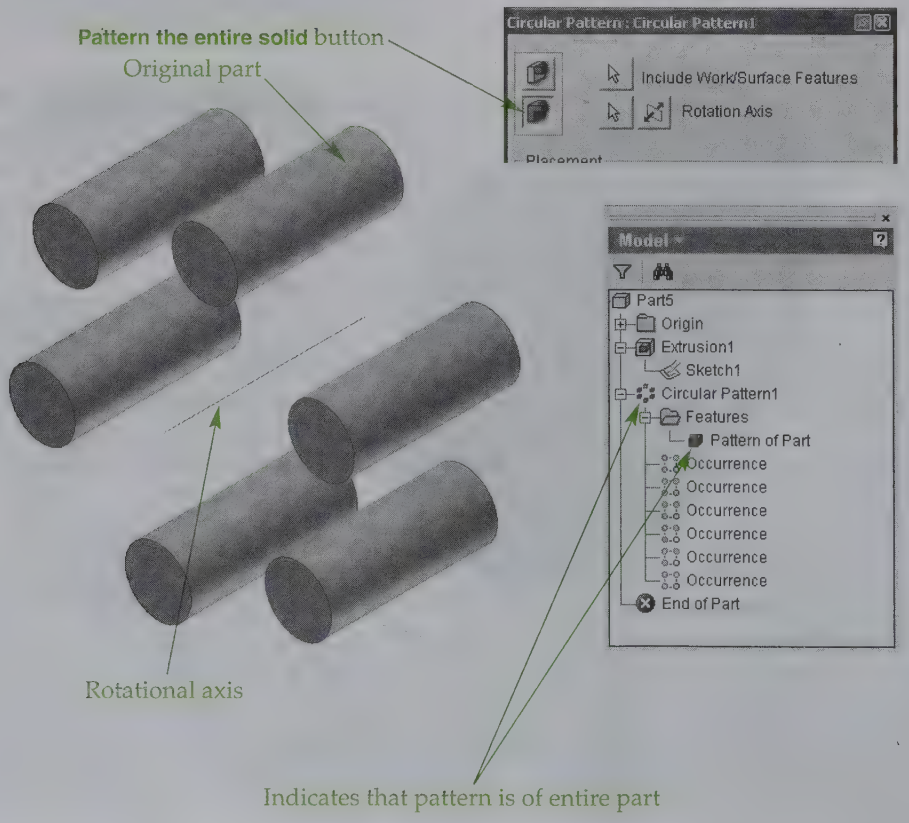


Figure 7-19.
The pattern direction arrow and pattern preview.

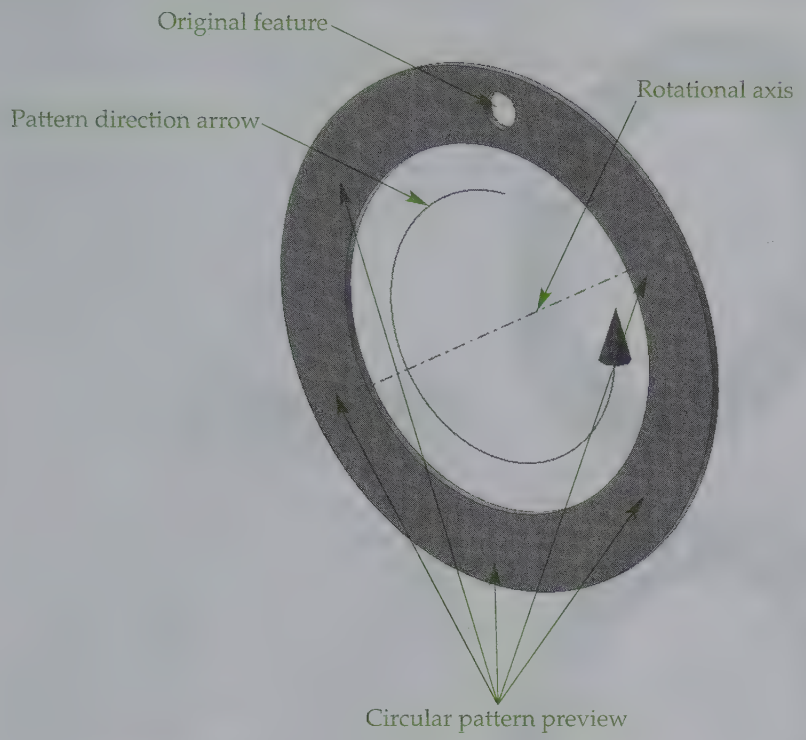
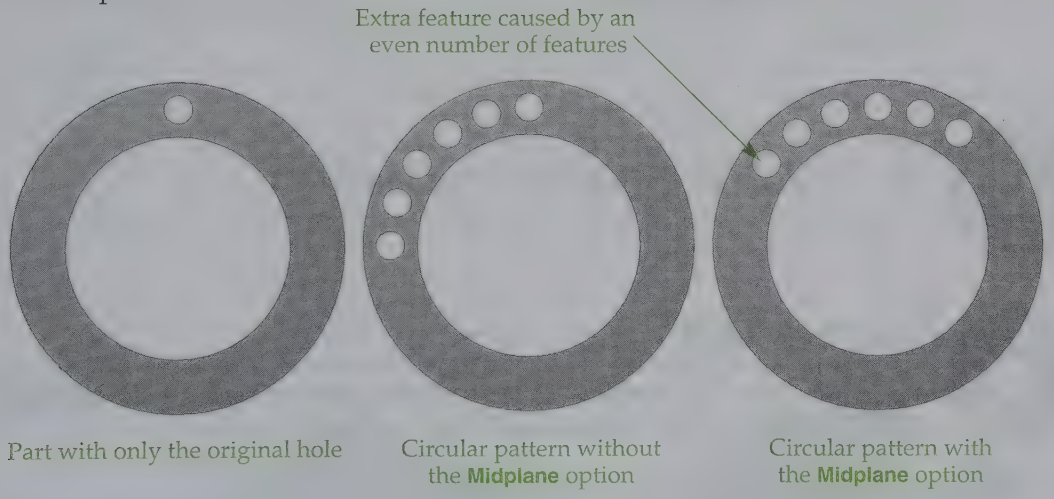


Figure 7-20.
Compare the locations of the patterned features when the **Midplane** direction option is and is not selected in these circular patterns. Selecting the **Flip** option will place the extra feature on the other side of the pattern.

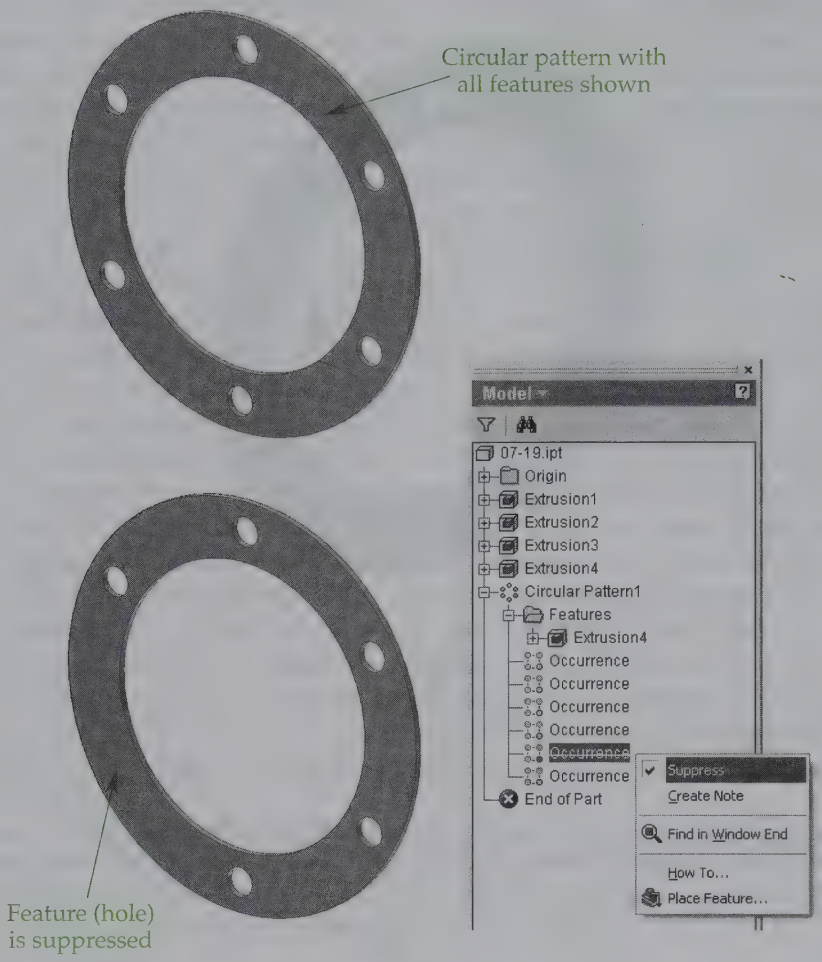


present as a result of the pattern operation, including the parent feature. To suppress a circular pattern feature, right-click the desired occurrence and pick the **Suppress** menu option. See Figure 7-21.

NOTE

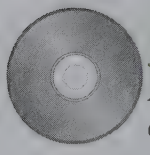
You cannot suppress the parent feature occurrence, but you can suppress the actual parent feature by right-clicking on the feature and selecting the **Suppress Feature** pop-up menu option.

Figure 7-21.
The effects of suppressing an occurrence of a circular pattern.





Exercise 7-3 Complete the Exercise on the Student CD.



Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. Describe feature patterns and give an example of these features.
2. When can pattern features be created, and when are features typically patterned?
3. How do you access the pattern feature tools?
4. Briefly discuss the restrictions concerning the patterning of placed features.
5. Describe a *mirrored feature* and give an example of its use.
6. What is a *mirror plane* and what can you choose as a mirror plane?
7. Discuss *rectangular feature patterns* and give an example of their use.
8. Describe what you get when you choose a path direction.
9. Briefly discuss *circular feature patterns* and give an example of their use.
10. Identify the *axis of rotation* and at least two features that can be used.

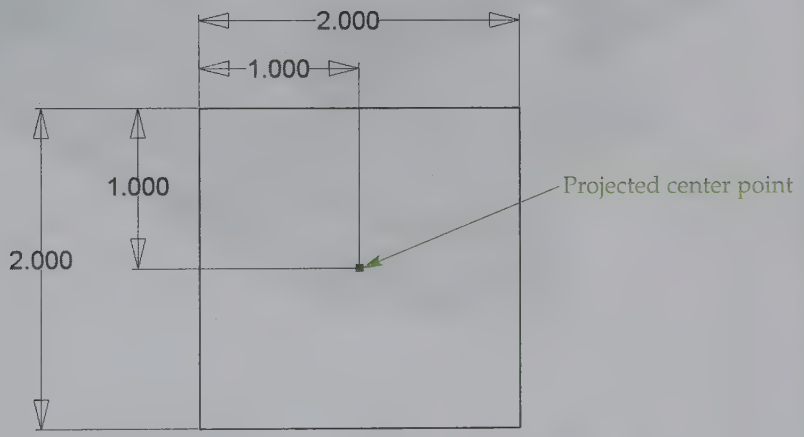
Problems

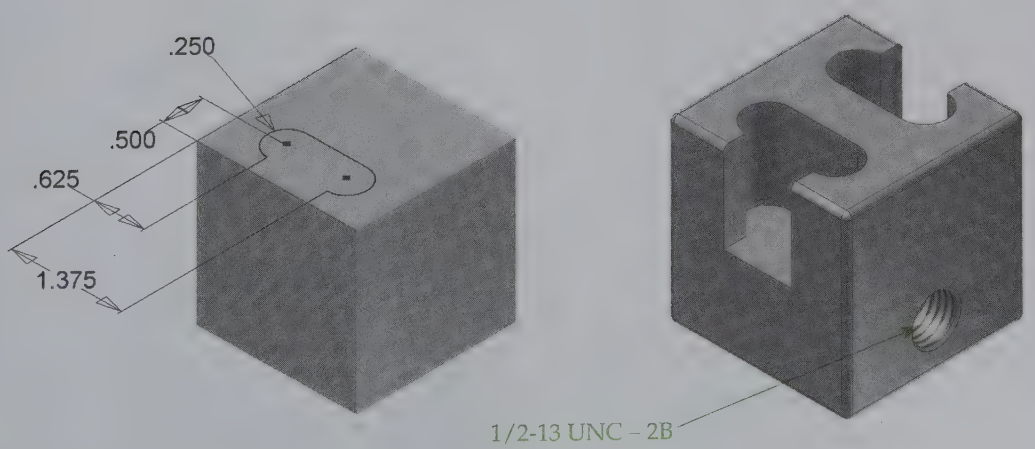
Instructions:

- Follow the specific instructions provided for each problem.

1. **Name:** Support
   Units: Inch
   Template: PART-in.ipt
   Material: Default
   Color: Default
   Specific Instructions:
   - Begin a new inch part file.
   - Create the sketch shown on the XY plane.
   - Extrude the sketch 2″ midplane.
   - Create the sketch shown on the top face.
   - Cut-extrude the sketch 1″ in a negative direction, as shown.
   - Mirror the cut extrusion using the XY Plane located under the **Origin** folder, as the **Mirror Plane**.
   - Add the specified hole .5″ from the bottom edge and 1″ from the vertical edge, as shown.
   - Place a .028625″ chamfer at the hole edges.
   - Place a .0625″ round on Extrusion1.

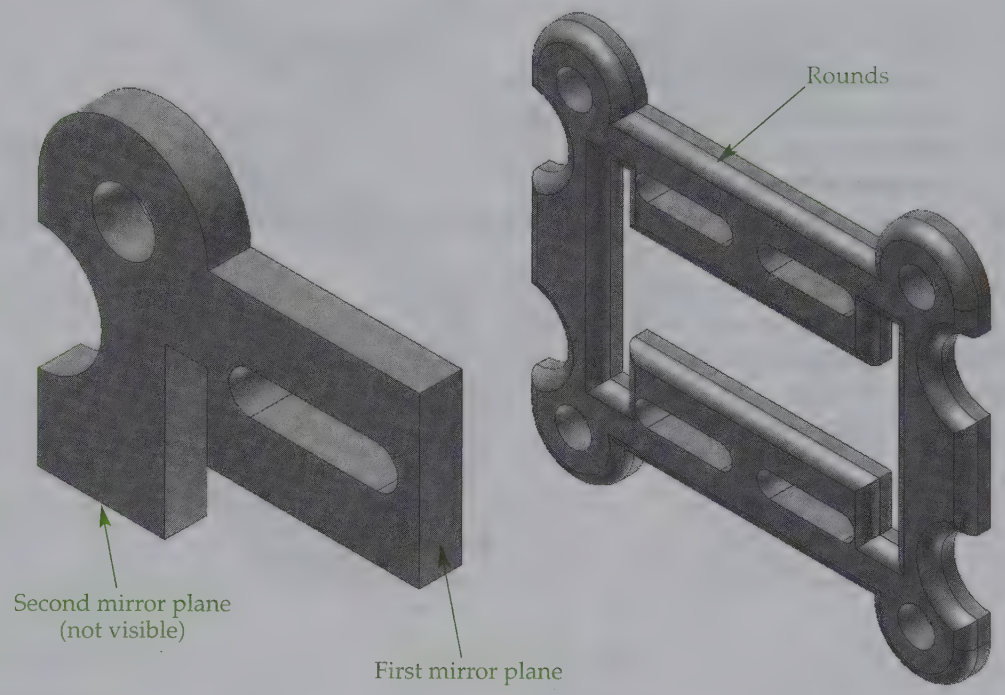
   The final part should look like the part shown.
   Save As: P7-1.ipt





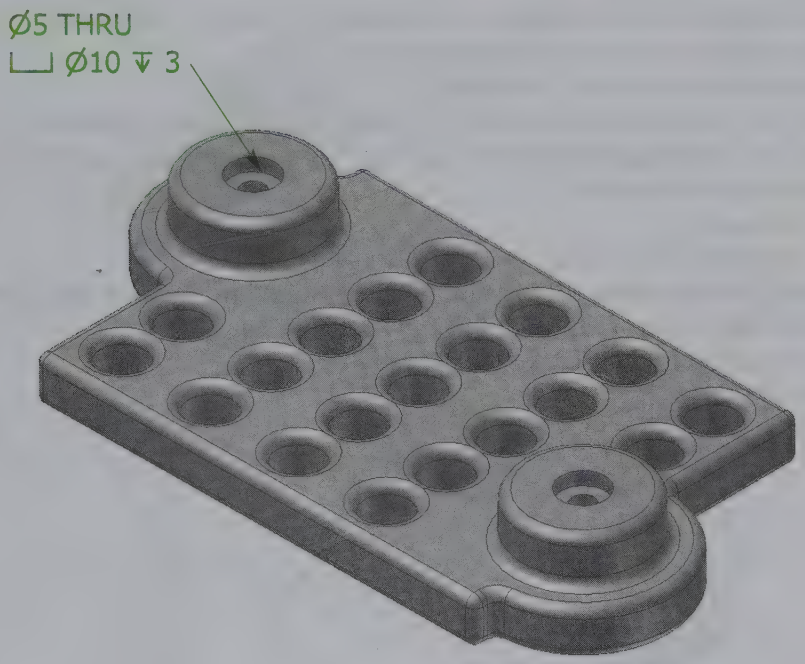
2. **Name:** Cover Plate
Units: Metric or Inch
Template: PART-in.ipt or PART-mm.ipt
Material: Your choice
Color: Your choice
Specific Instructions:
- Create features similar to the ones shown, using your own specifications.
- Mirror the features twice to create the part shown.
- Place rounds on the 2 loops shown.

Save As: P7-2.ipt



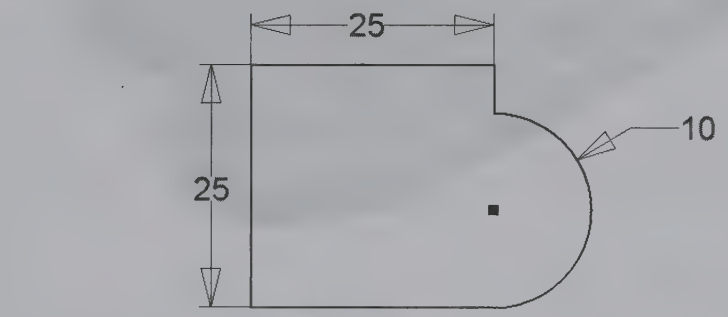
3. **Name:** Selector Bracket
   Units: Metric
   Template: PART-mm.ipt
   Material: Your choice
   Color: Your choice
   Specific Instructions:
   - Open EX7-2.ipt and save a copy as P7-3.ipt.
   - Close EX7-2.ipt without saving and open P7-3.ipt.
   - Sketch the Ø27.5 mm circles shown on the top face, and extrude them 10 mm.
   - Suppress 6 of the rectangular pattern holes, as shown.
   - Use the **Fillet** tool to fillet and round All Fillets and All Rounds, 2 mm.
   - Add the specified holes.
   - Open the **Properties** dialog box for the model and enter SELECTOR BRACKET in the **Title** edit box, and P7-3 in the **Part Number** edit box.

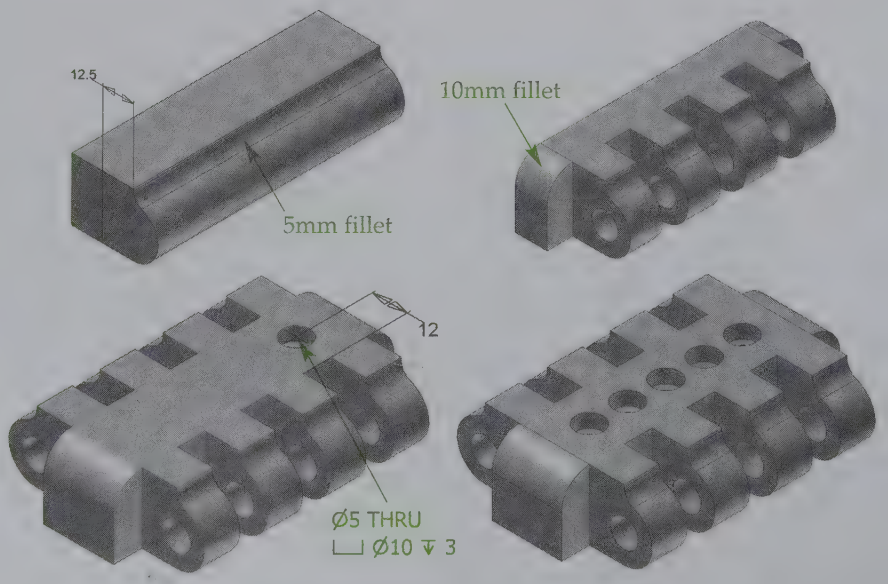
   Save As: P7-3.ipt



4. **Name:** Slide Bar Hinge Connector
Units: Metric
Template: PART-mm.ipt
Material: Titanium
Color: As Material
Specific Instructions:
- Begin a new metric part file.
- Create the sketch shown on the XY plane.
- Extrude the sketch 100 mm in a positive direction.
- Place a 5 mm fillet as shown.
- Create the sketch shown on the specified face.
- Cut-extrude the sketch 10 mm in a negative direction.
- Use the **Rectangular Pattern** tool to pattern the extrusion as shown, with the following information:
  - **Count:** 5
  - **Spacing:** 22.5 mm
- Add a 10 mm hole through all the feature geometry, as shown.
- Place a 10 mm fillet on the edges shown.
- Mirror the entire solid as shown.
- Add the specified hole, vertically constrained to the edge shown.
- Use the **Rectangular Pattern** tool to pattern the hole as shown, with the following information:
  - **Count:** 5
  - **Spacing:** 13.75 mm
- Open the **Properties** dialog box for the model and enter SLIDE BAR HINGE C. in the **Title** edit box, and P7-4 in the **Part Number** edit box.

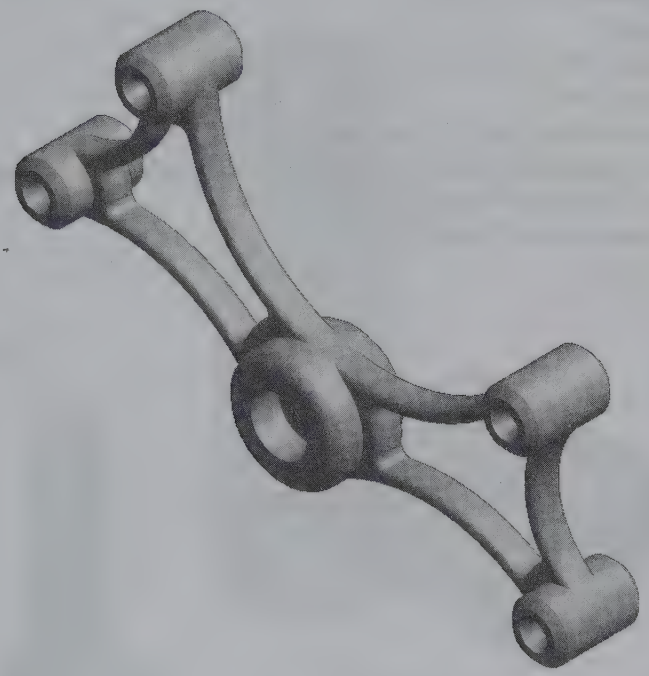
The final part should look like the part shown.
Save As: P7-4.ipt.





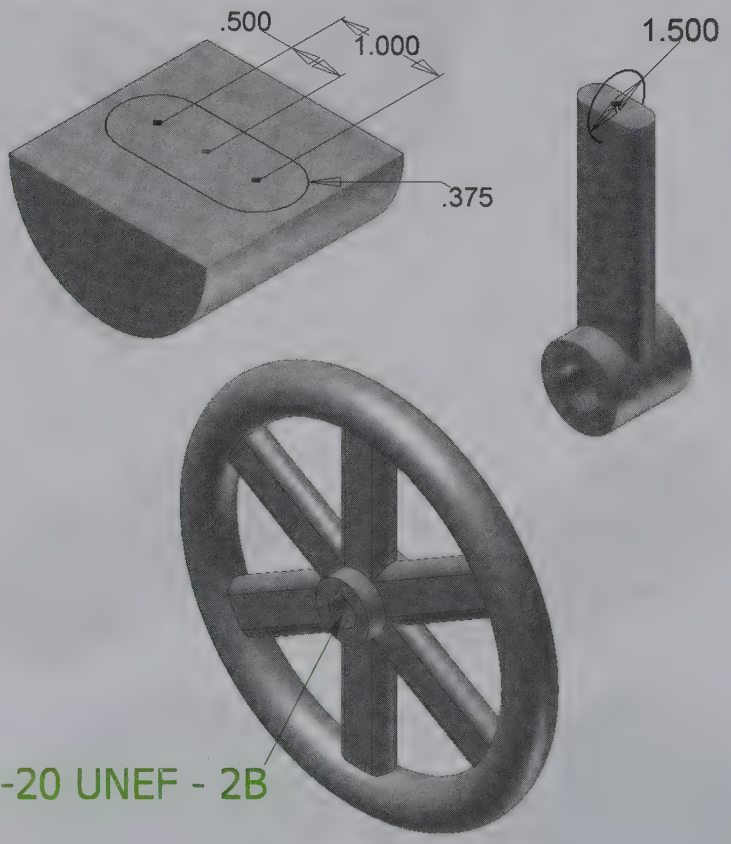
5. **Name:** Support Bracket
Units: Inch
Template: PART-in.ipt
Material: Aluminum-6061
Color: As Material
Specific Instructions:
   - Open P6-4 and save a copy as P7-5.
   - Close P7-4 without saving and open P7-5.
   - Use the **Circular Pattern** tool to pattern the entire solid as shown, with the following information:
     - **Rotation Axis:** Z axis
     - **Count:** 2
     - **Position Method:** Incremental
     - **Angle:** 135°

   The final part should look like the part shown.
   Save As: P7-5.ipt.



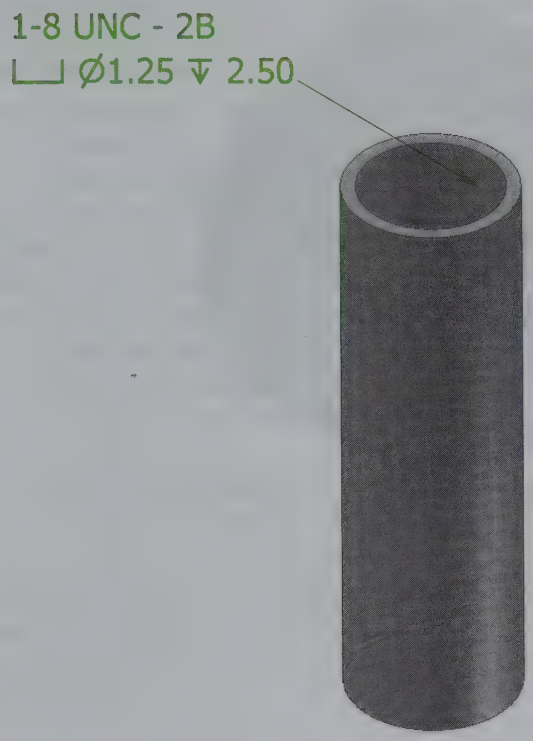
6. **Name:** Wheel
Units: Inch
Template: PART-in.ipt
Material: Steel, Mild
Color: Orange (Dark)
Specific Instructions:
- Begin a new inch part file.
- Sketch a ∅2″ circle around the projected center point, on the XY plane.
- Extrude the sketch 2″, midplane.
- Sketch the geometry shown on the XZ plane. You may need to use the **Slice Graphics** option to clarify the sketch environment.
- Extrude the sketch 6″ in a positive direction.
- Sketch the circle shown on the YZ plane.
- Fully revolve the sketch around the Z axis.
- Use the **Circular Pattern** tool to pattern Extrusion2 as shown, with the following information:
  - **Rotation Axis:** Z axis
  - **Count:** 6
  - **Position Method:** Fitted
  - **Angle:** 360°
- Add the specified hole to Extrusion 1.
- Open the **Properties** dialog box for the model and enter WHEEL in the **Title** edit box, and P7-6 in the **Part Number** edit box.

The final project should look like the part shown.
Save As: P7-6.ipt.



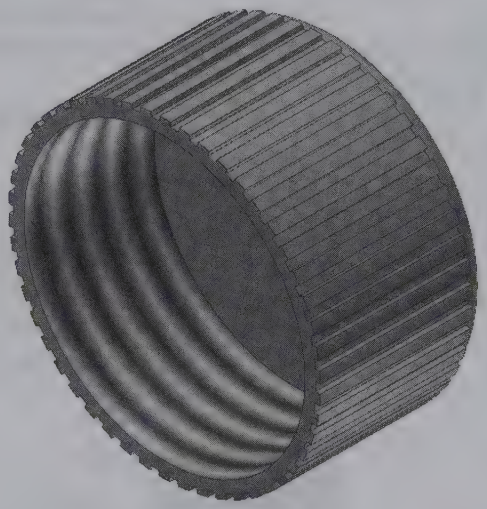
7. **Name:** Hub
Units: Inch
Template: PART-in.ipt
Material: Default
Color: Default
Specific Instructions:
- Begin a new inch part file.
- Sketch a circle ∅1.5″ circle on the XZ plane.
- Extrude the sketch 5″ in a positive direction.
- Add the specified hole as shown.
- Open the **Properties** dialog box for the model and enter PROBLEM 7-7 in the **Title** edit box, and P7-7 in the **Part Number** edit box.

Save As: P7-7.ipt.



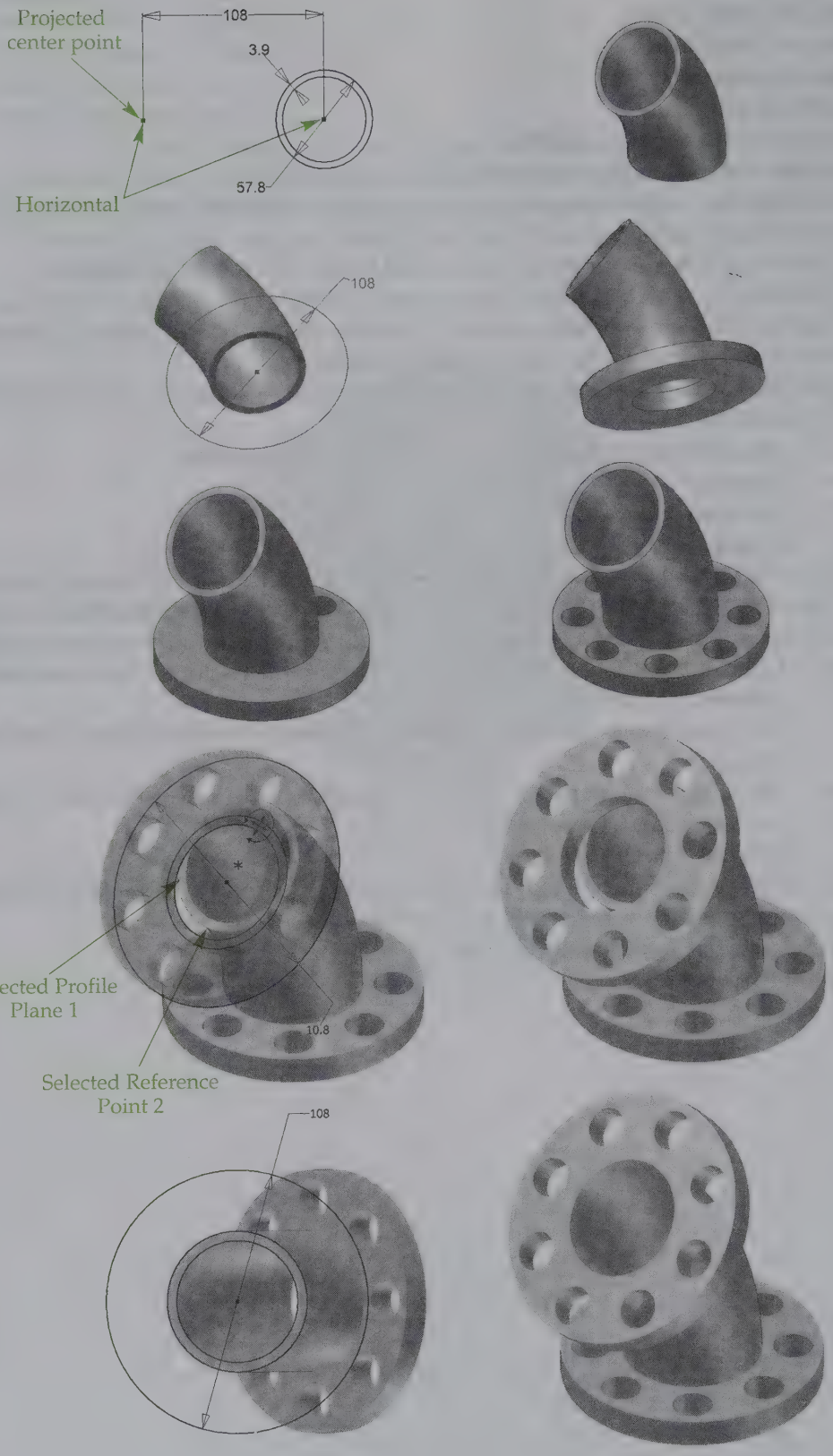
8. **Name:** Bottle Cap
Units: Inch or Metric
Template: PART-in.ipt or PART-mm.ipt
Material: Default or your choice
Color: Default or your choice
Specific Instructions:
   - Develop a cap for the bottle you created in Problem 6-8.
   - Use the **Circular Pattern** tool to apply straight knurls to the cap as shown. Sketch the parent knurl feature from one of the reference planes located in the Origin folder, and extrude through the cap. Then apply the circular pattern, followed by the hole.

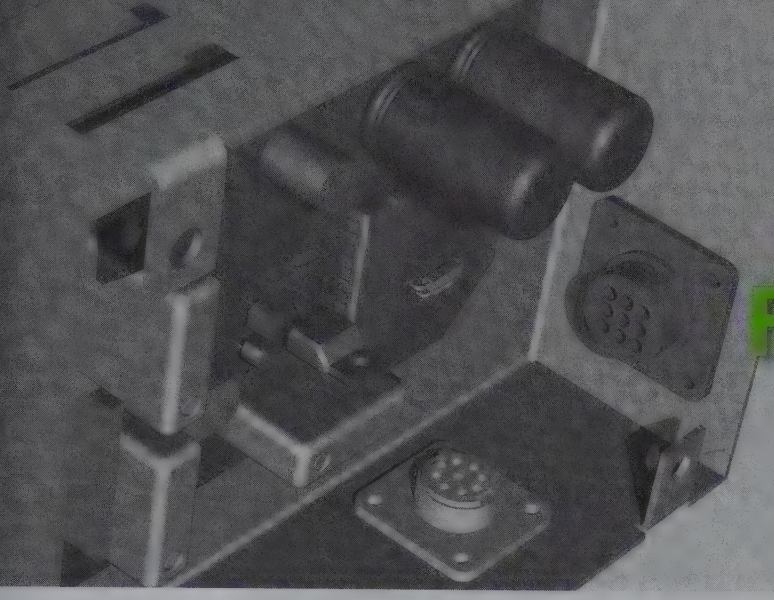
Save As: P7-8.ipt.



9. **Name:** 45° Elbow
Units: Metric
Template: PART-mm.ipt
Material: Steel, Mild
Color: As Material
Specific Instructions:
   - Begin a new metric part file.
   - Create the sketch shown on the XZ plane.
   - Revolve the profile formed by both circles, around the Z axis, an angle of 45°, in a positive direction.
   - Sketch a ∅108 mm circle on the bottom face of the revolved feature.
   - Extrude the sketch 11.5 mm away from the revolution as shown.
   - Add a ∅16.66 mm hole as shown. The hole should be horizontal to, and 41.5 mm from, the center of the extruded cylinder.
   - Use the **Circular Pattern** tool to pattern the hole as shown, with the following information:
     - **Rotation Axis:** Select the curved Extrustion1 surface
     - **Count:** 8
     - **Position Method:** Fitted
     - **Angle:** 360°
   - Copy Extrusion1 to the top face of the revolved feature by dragging Extrusion1 from the browser into the graphics window. Use the **Paste Features** dialog box and the following information to paste the extrusion and the pattern of holes:
     - Choose **Dependent** from the paste features drop-down list to include the pattern of holes.
     - Highlight Profile Plane1 in the **Name** list and select the top face of the revolution.
     - Highlight Reference Point2 in the **Name** list and select the outer edge of the top face of the revolution.
     - Pick the **Finish** button.
   - Edit the sketch of the pasted extrusion and use Concentric, followed by Tangent geometric constraints to fully constrain the sketch.

   The final project should look like the part shown.
   Save As: P7-9.ipt.


Projected
center point
108
3.9
57.8
Horizontal
108
Selected Profile
Plane 1
10.8
Selected Reference
Point 2
108



CHAPTER 8

Work Features

Learning Objectives

After completing this chapter, you will be able to do the following:

- Create work planes.
- Establish work axes.
- Use ungrounded and grounded work points.

Often while developing a part model, you may not have sufficient geometry or areas on which to sketch, dimension and constrain, and to build features. As a result, Inventor provides you with work features. ***Work features*** are similar to construction lines and surfaces, and allow you to create reference elements anywhere in space to help position and generate additional features. Using work features ensures that you can create complex parametric shapes. Work features provide the same function as the default work planes, axes, and center point. However, user-defined work features allow you to specify the location of planes, axes, and points. See Figure 8-1.

Work features can be created anytime during part model development. However, most work features, especially axes and points, are added when at least one feature currently exists. The easiest way to create work features without using existing geometry is to reference the default work features from the **Origin** folder of the browser bar. Figure 8-2 shows how several work planes can be generated by referencing the default work planes.

Using Work Planes

A ***work plane*** is a flat reference surface that can be located anywhere in space. Though you may see only a portion of a work plane in the graphics window, work planes are infinite. This means that they extend, or cover, the entire plane of the location where they are placed. As you work with Inventor and continue to develop parts and assemblies, you will find many uses for work planes. Generally, you can use work planes to create sketches when no default reference planes or feature faces are available, define another work feature, add a termination plane for creating features such as extrusions, and create constraint planes for assemblies.

Work planes appear semitransparent in the graphics window. See Figure 8-3. If at any time you want to turn off the visibility or hide a work plane, right-click on the

Figure 8-1.
Examples of work planes, work axes, and work points.

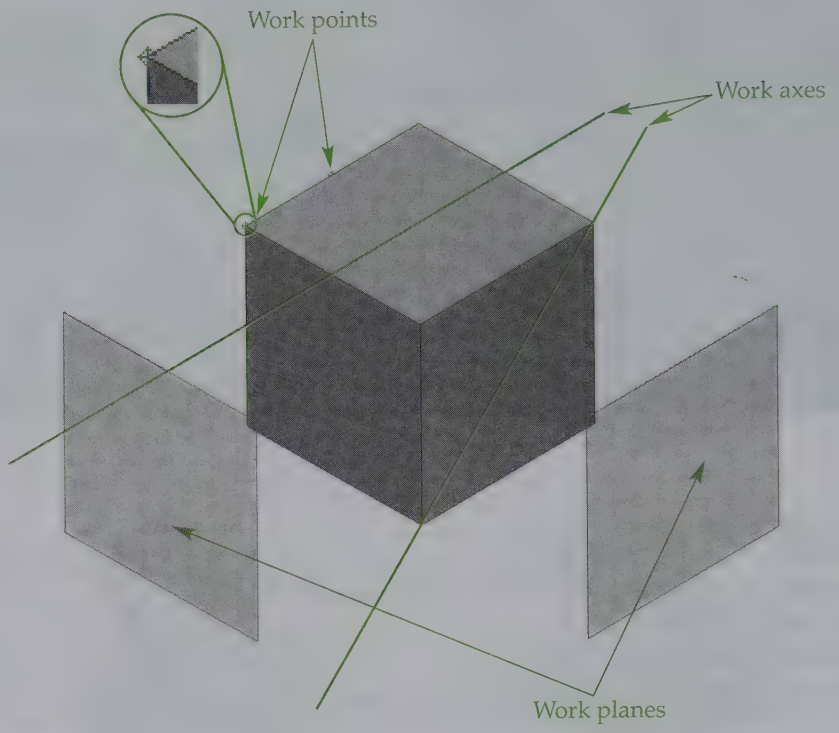
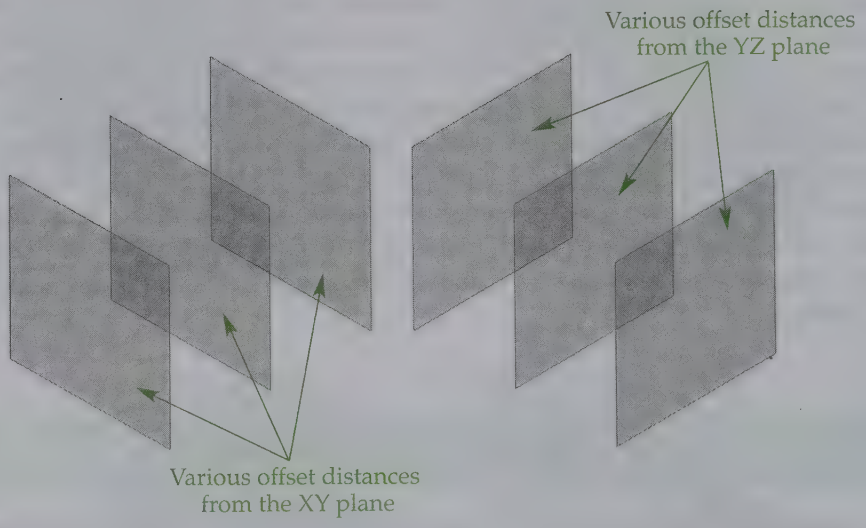
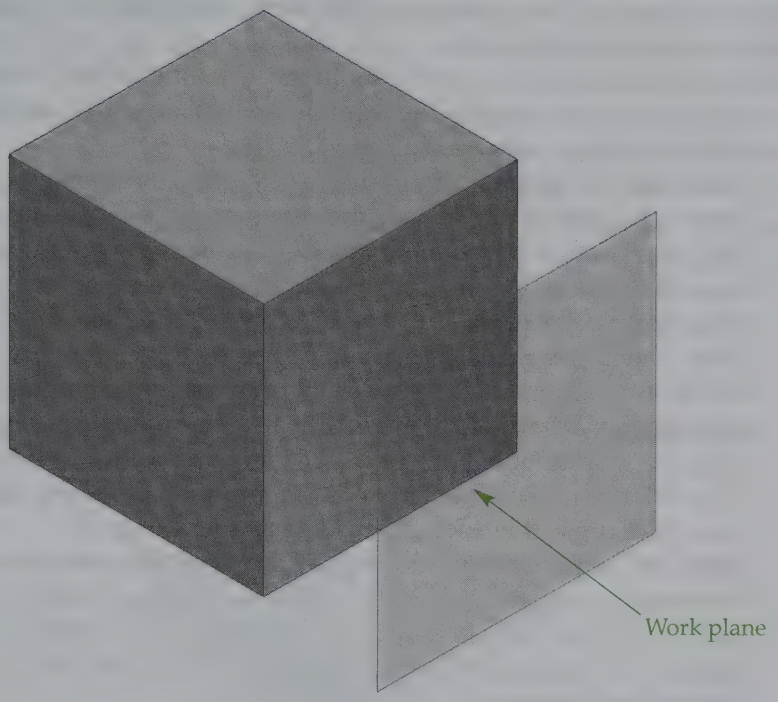


Figure 8-2.
Creating work features by referencing existing feature geometry.



work plane in the browser and deselect the **Visibility** pop-up menu option, or pick the work plane in the graphics window, right-click, and deselect the **Visibility** pop-up menu option. You can also move and manipulate the displayed size of a work plane by hovering your cursor over the edge until you see the **Move** icon. See Figure 8-4. Then, hold down the left mouse button and drag the work plane to the desired location. When you finish moving the work plane, release the mouse button. To change the displayed size of the work plane, move your cursor over one of the work plane corners until you see the **Resize** icon, and hold down the left mouse button and drag

Figure 8-3.
A visible work plane.



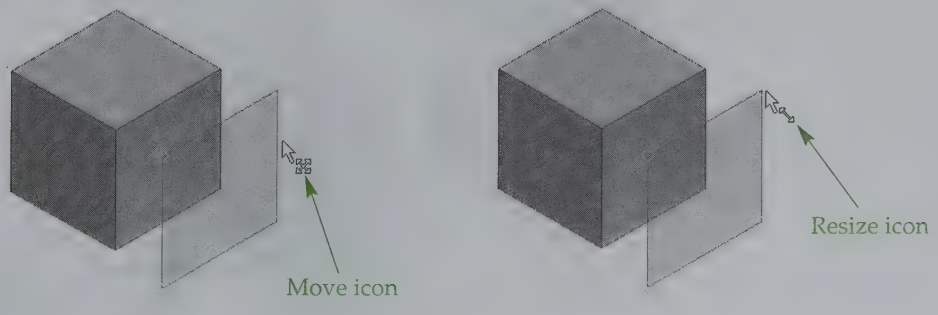
the corner of the work plane to the desired location and size. When you finish resizing the work plane, release the mouse button. See Figure 8-4.

NOTE

Moving or resizing the work plane display does not change the specified location, constraints, or area of the work feature. To physically relocate the work plane in space, you must right-click on the work plane in the browser and select the **Redefine Feature** menu option, or pick the work plane in the graphics window, right-click, and select the **Redefine Feature** menu option. You can then recreate the work plane.

Work planes are created using the **Work Plane** tool, accessed by picking the **Work Plane** button on the **Part Features** panel bar; picking the **Work Plane** button on the **Part Features** toolbar; or pressing the right bracket (]) key on your keyboard.

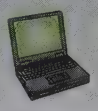
Figure 8-4.
Notice the icon shape when moving and resizing a work plane.



When you access the **Work Plane** tool or any other work feature, no dialog box opens. Instead, you define where you want to place the work plane by selecting existing reference geometry. Once you access the **Work Plane** tool, use one of the following techniques to create a work plane:

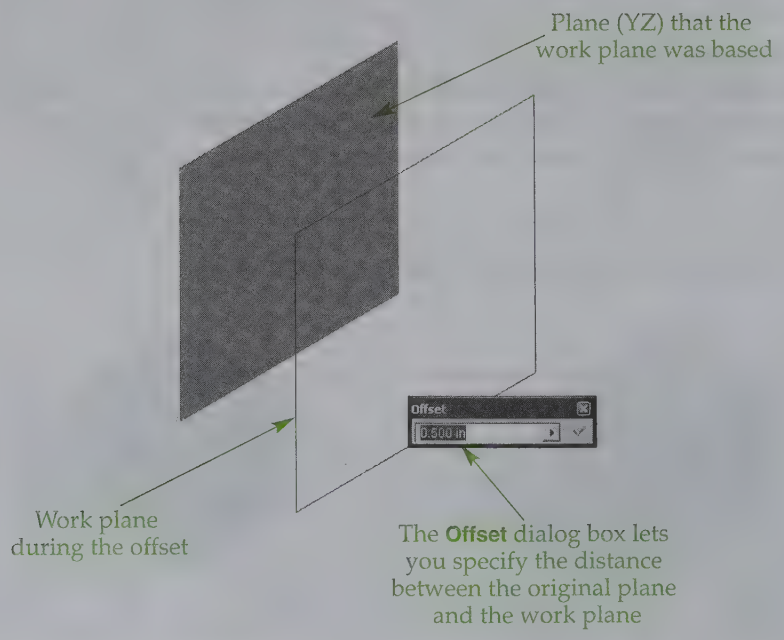
- **Using default reference work features.** Use one of the following options to create a work plane using the default reference work features located in the **Origin** folder of the browser:
  - **Select the YZ plane, XZ plane, or XY plane.** Selecting one of the default planes displays a preview of the plane and the work plane icon. To locate the plane, hold down the left mouse button and drag the work plane to the desired location. While you are dragging the work plane, the **Offset** dialog box appears. See Figure 8-5. Use the **Offset** dialog box to help place the work plane. Once you release the mouse button, you can enter a value in the edit box. Use a value of 0 to create the work plane at its current location.
  - **Select two available axes.** Selecting any pair of default reference work axes creates a work plane. For example, if you select the X axis and the Y axis, an XY work plane is created. You can accomplish the same task by referencing one of the default work planes, such as the XY plane.
  - **Select the center point and one of the available axes or planes.** Selecting the center point and a default work axis or plane creates a work plane. For example, if you select center point and the X axis, an YZ plane is created. Again, you can accomplish the same task by referencing one of the default work planes, such as the YZ plane.

PROFESSIONAL TIP



Use the offsetting techniques described to offset a new plane from an existing plane.

Figure 8-5.
This work plane was offset from the default YZ plane.



- **From an existing feature face.** Use this option to define a work plane parallel to, and a specified distance from, a face. To create a work plane using an existing feature face, pick the face from which you want to offset the work plane. Then offset the work plane using the offsetting technique previously described. See Figure 8-6.
- **From two feature edges or axes.** This option generates a work plane coplanar to two edges or axes. To create a work plane by referencing two edges, pick the edges or axes where you want the work plane to be located. See Figure 8-7.
- **Using three existing points.** This option, often referred to as a three-point work plane, generates a work plane by referencing any three points, including feature edge endpoints, intersections, midpoints, and work points. To create a three-point work plane, pick three different points. See Figure 8-8.

Figure 8-6.
Creating a work plane at a specified location by referencing an existing feature face.

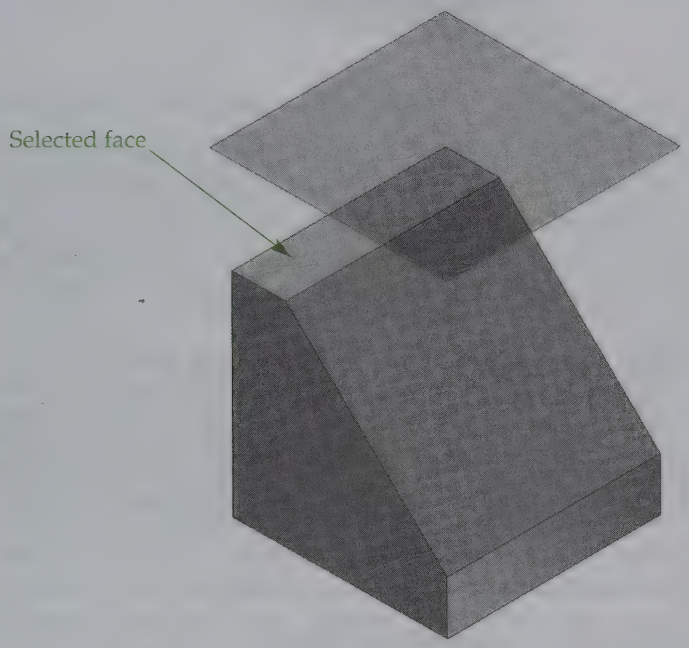


Figure 8-7.
Creating a work plane coplanar to two edges or axes.

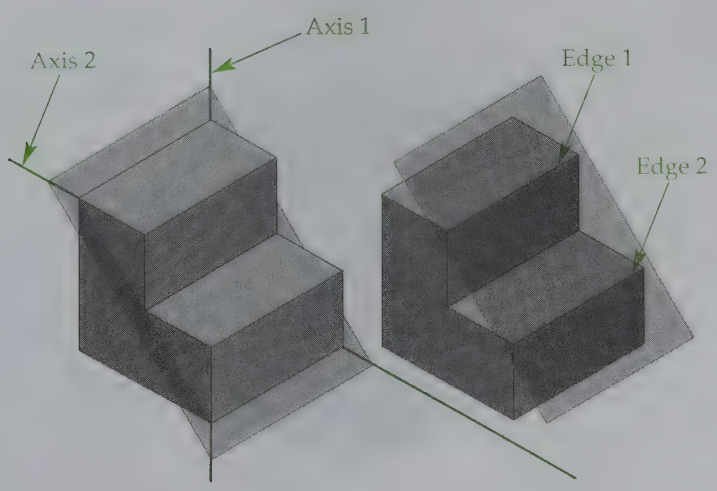
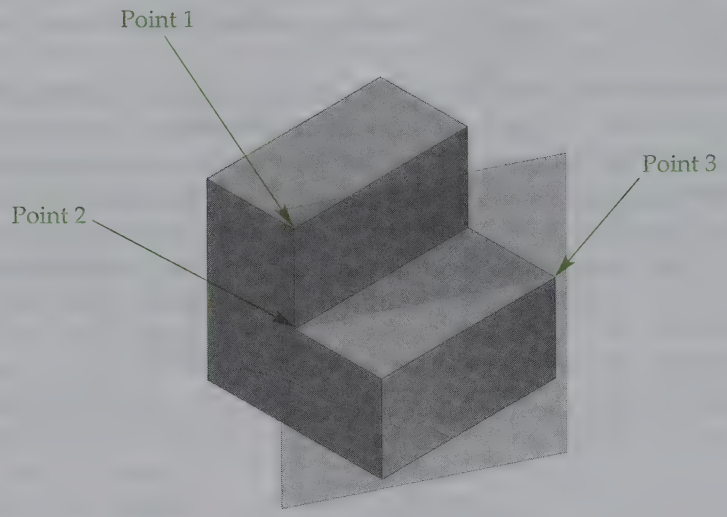


Figure 8-8.
Creating a work plane by selecting three points.



- **From an edge or axis and a point.** Use this option to define a work plane perpendicular to the selected edge, through a specified point. To create a work plane by referencing an edge or axis and a point, pick the edge or axis, and then pick any point including feature edge endpoints, intersections, midpoints, center points, and work points. Or, select the point, followed by the edge or axis. See Figure 8-9.
- **From a face or plane and a point.** This option allows you to define a work plane parallel to a selected face or plane through a specified point. To create a work plane by referencing a face or plane and a point, pick the face and then pick any point including a feature edge endpoint, intersection, midpoints, center point, and work point. Or, select the point, followed by the face. See Figure 8-10.

Figure 8-9.
Creating a work plane by selecting an edge or axis and a point.

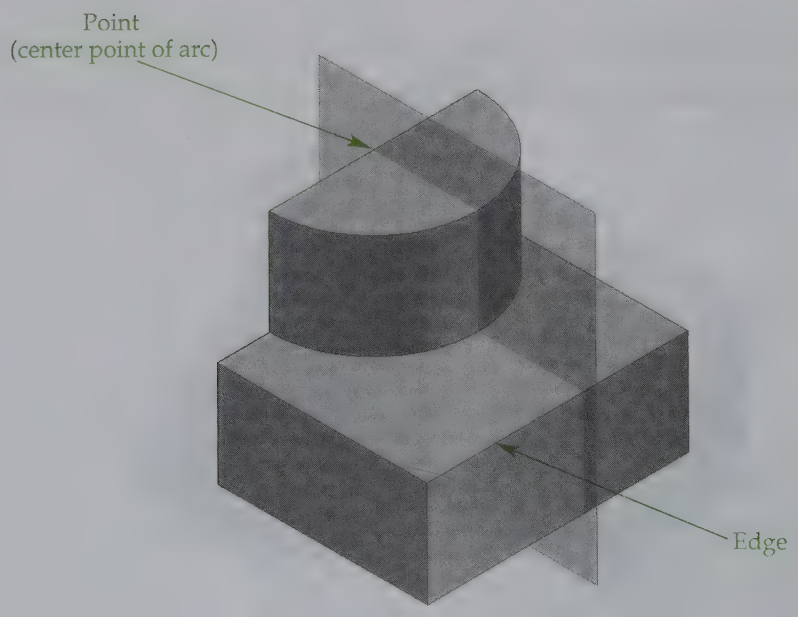
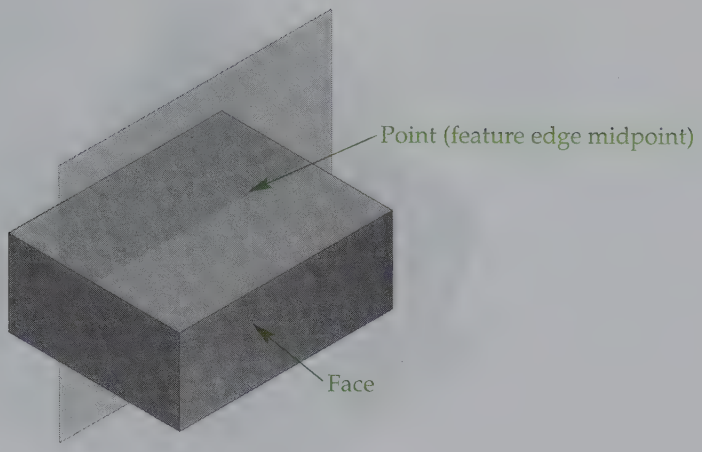


Figure 8-10.
Creating a work plane by selecting a face and a point.



- **Using a tangent face and an edge or axis.** This option defines a work plane tangent to the selected curved feature, such as a cylinder or fillet, and coplanar to the selected edge or axis. To create a work plane by referencing a tangent face and an edge or axis, pick the face and then pick the edge or axis. Or, select the edge or axis, followed by the face. See Figure 8-11.
- **Using a tangent face and a planar face.** This option defines a work plane tangent to the selected curved feature, such as a cylinder or fillet, and parallel to a different face. To create a work plane by referencing a tangent face and a planar face, pick the tangent face and then pick the other face. Or, select the planar face, followed by the tangent face. See Figure 8-12.
- **Angled from a planar edge or axis and a face.** Use this option to define a work plane at an angle to the selected face, through a specified edge or axis. To create a work plane by referencing a face and an edge or axis, pick the face and then pick an edge or axis. Or, select the edge or axis, followed by the face. See Figure 8-13. Once you pick an edge or axis and a face, the **Angle** edit box will appear, allowing you to specify the angle of the work plane. The selected edge acts as the angle pivot point.

Figure 8-11.
Creating a work plane by selecting a tangent face and an edge or axis.

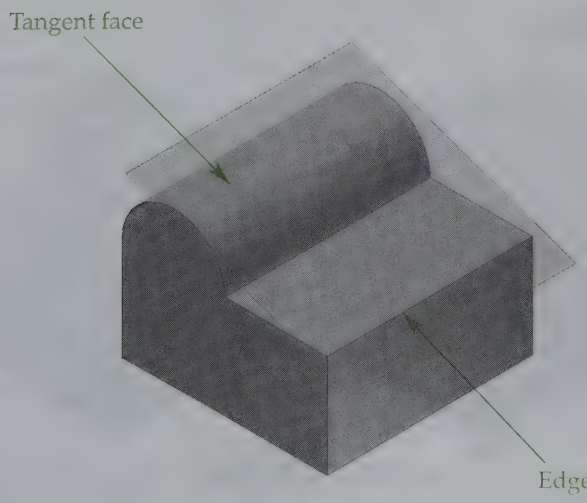
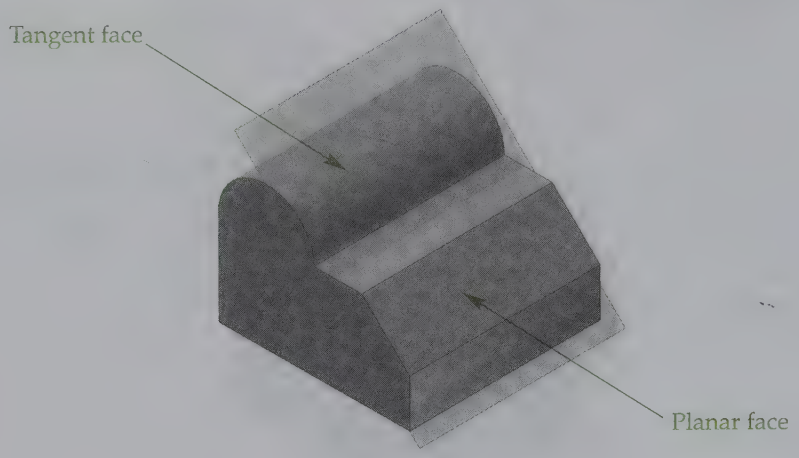


Figure 8-12.
Creating a work plane by selecting a tangent face and a planar face.



NOTE

The **Angle** edit box defines the angle of the work plane. This means that you can select any face parallel to the selected edge and create the same work plane by entering different angles.

- **Tangent to a cylinder.** This option defines a work plane tangent to the selected curved feature, such as a cylinder or fillet. To create a work plane by referencing a tangent face, another feature or sketch must be available for selection. If the cylinder was created by referencing the center point and you want the work plane to be parallel to a default work plane, you can select the tangent feature and one of the default reference work planes. See Figure 8-14. If you want to create a work plane tangent to a cylinder, but cannot reference one of the default work planes, or you do not want the work plane to be parallel to one

Figure 8-13.
Creating a work plane by selecting a face and an edge or axis.

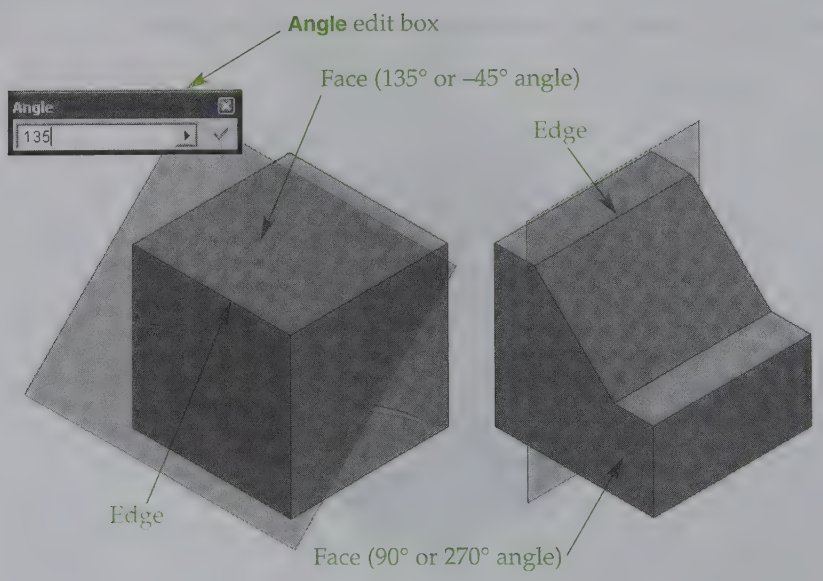
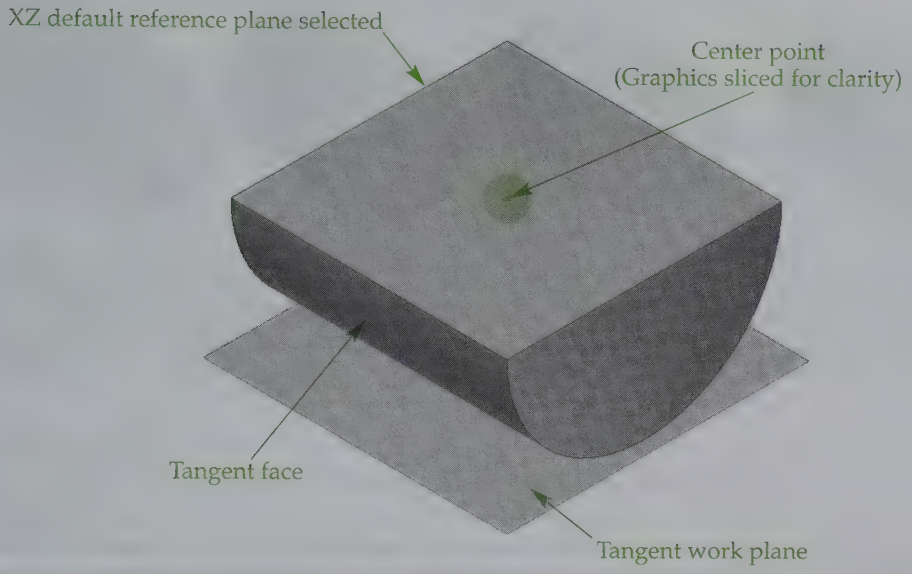


Figure 8-14.
Creating a work plane tangent to a cylinder.



of the default work planes, you must access alternative geometry. One option is to create a sketch, using construction lines on one of the cylinder faces. Once you sketch the line or lines, pick the tangent point on the cylinder edge and then select the sketched line, or pick the sketched line followed by the tangent point of the cylinder edge. See Figure 8-15.

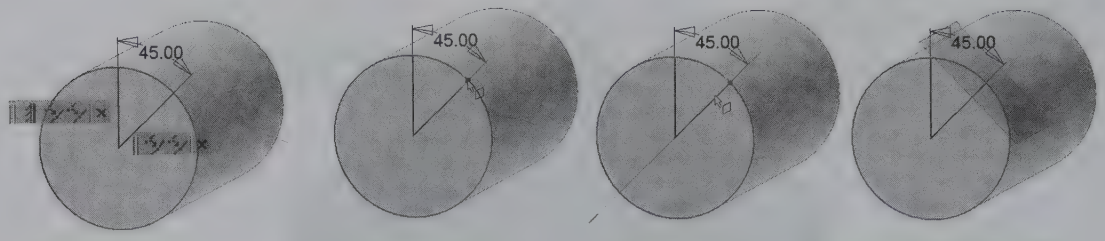
PROFESSIONAL TIP



The construction line endpoints must be coincident to the tangent edge and axis of the cylinder. To fully constrain the sketch, generate lines from the center point of the cylinder, and apply any needed dimensions or geometric constraints. When finished using the sketch, you may want to hide the sketch geometry by right-clicking on the sketch in the browser, and deselect the **Visibility** menu option.

- **Bisecting two existing planes or faces.** Use this option to define a work plane parallel to, and half the distance between, two existing work planes or feature faces. To create a bisecting work plane, pick either faces or planes. See Figure 8-16.
- **Using a curve and a point on the curve.** This option generates a work plane perpendicular to a curve at a selected point on the curve, such as a line, circle, arc, spline, or ellipse. Then, pick a point on the curve; such as an endpoint, work point, center point, or midpoint, where you want the work plane to be located. See Figure 8-17. Another example of creating a work plane perpendicular to a curve is shown in Figure 8-15. This example uses a sketch on an extruded cylinder face. This technique can be used to crate a work plane tangent to a cylinder, when you cannot reference one of the default work planes, or you do not want the work plane to be parallel to one of the default work planes.

Figure 8-15.
Sketched construction lines used to create a work plane tangent to a cylinder.



Create a sketch on the end of the cylinder

Start building the work plane by picking the end of the angled line

Pick the line

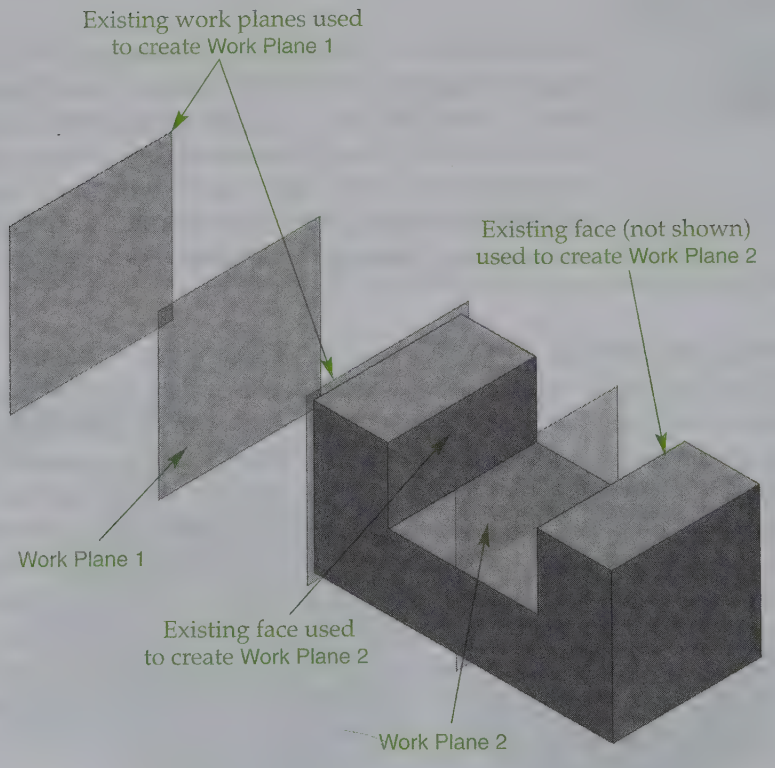
The work plane is created perpendicular to the selected sketch line, at the selected endpoint

NOTE



Once you add any of the work planes previously described, you can offset another work plane. To accomplish this task, access the **Work Plane** tool, pick the existing work plane, and apply the offsetting technique described.

Figure 8-16.
Creating a work plane that bisects two existing planes or faces.



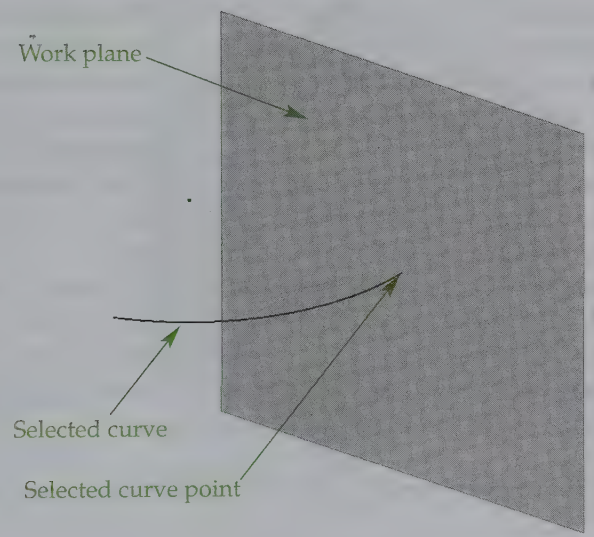
Exercise 8-1 Complete the Exercise on the Student CD.

Exercise 8-2 Complete the Exercise on the Student CD.

Exercise 8-3 Complete the Exercise on the Student CD.

Exercise 8-4 Complete the Exercise on the Student CD.

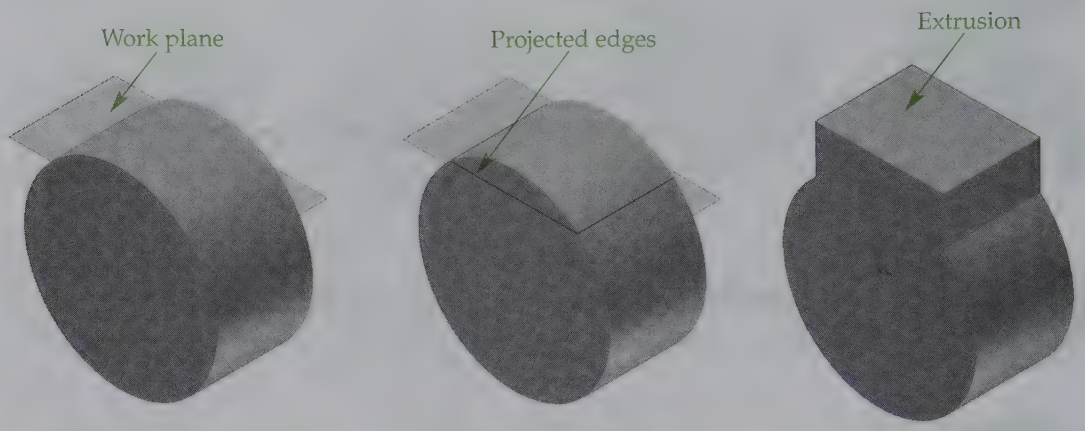
Figure 8-17.
Creating a work plane perpendicular to a curve's plane, at a curve point.



Projecting Cut Edges

When a surface, such as a work plane, cuts through a feature it can be difficult to reference the intersection of nonplanar feature faces and the work plane for sketch development. Figure 8-18 shows a sketch opened on a work plane that cuts through a portion of an extruded cylinder. The work plane cuts through curved feature faces that cannot be projected using the **Project Geometry** tool. Use the **Project Cut Edges** tool to project the intersection of the work plane and the feature faces onto the sketch. The **Project Cut Edges** tool is located in the default **Project Geometry** flyout button. When you select the **Project Cut Edges** button, all feature faces cut by the plane are automatically projected onto the sketch as edges; there is no need to individually pick the intersection.

Figure 8-18.
Projecting sketch geometry created by the intersection of feature faces and a work plane.

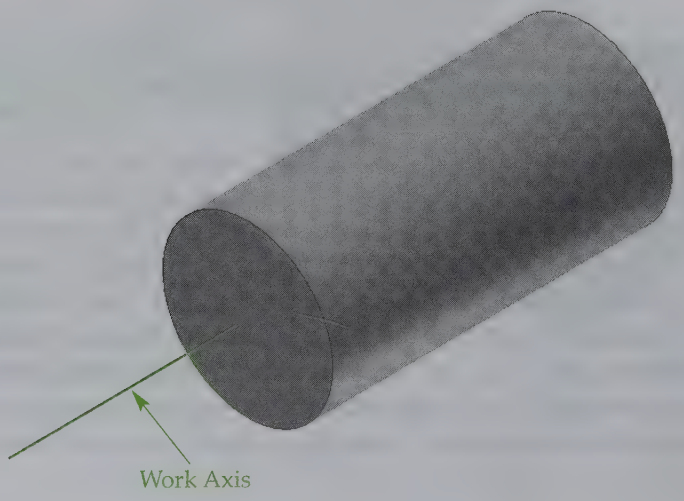


Using Work Axes

A ***work axis*** is a parametric reference line that can be located anywhere in space. Like work planes, you see only a portion of a work axis in the graphics window even though work axes are infinite and extend the entire length of the location where they are placed. As you work with Inventor and continue to develop parts and assemblies, you will find numerous uses for work axes. Generally, you may want to use work axes to create work axes when no default reference axes, feature edges, or points are available, create an axis for revolved and coiled features, create a direction reference line for rectangular patterns, or a rotation axis for circular patterns, create constraint axes for assemblies, and help generate additional work features.

Work axes are displayed as thin lines in the graphics window. They extend past referenced features and may be partially hidden. See Figure 8-19. If at any time you want to turn off the visibility of, or hide, a work axis, right-click the work axis in the browser and deselect the **Visibility** menu option, or pick the work axis in the graphics window, right-click, and deselect the **Visibility** menu option.

Figure 8-19.
A visible work axis.



Work axes are created using the **Work Axis** tool, accessed by picking the **Work Axis** button on the **Part Features** panel bar; picking the **Work Axis** button on the **Part Features** toolbar; or pressing the [/] key on your keyboard.

When you access the **Work Axis** tool, you define where you want to place the work axis by selecting existing reference geometry. Once you access the **Work Axis** tool, use one of the following techniques to create a work axis:

- **Using default reference work features.** Though you can select the default work features located in the **Origin** folder of the browser to create a work axis, you can accomplish the same task by referencing one of the default work axes. For example, if you pick the center point and the YZ plane, an axis identical to the default X axis is created.
- **From two existing points.** Use this option to generate a work axis by referencing any two points, including feature edge endpoints, intersections, midpoints, and work points. To create a work axis by referencing points, pick two different points. See Figure 8-20.
- **Using two intersecting work planes or feature faces.** This option generates a work axis coplanar to, and at the intersection of, two work planes or faces. To create a work axis by referencing two work planes or faces, pick the work planes or faces where you want the work axis to be located. See Figure 8-21.

Figure 8-20.
Creating a work axis by selecting two points.

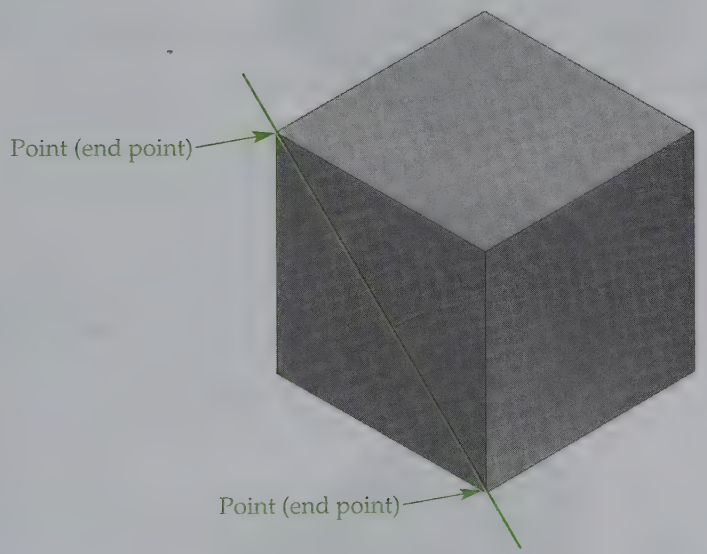
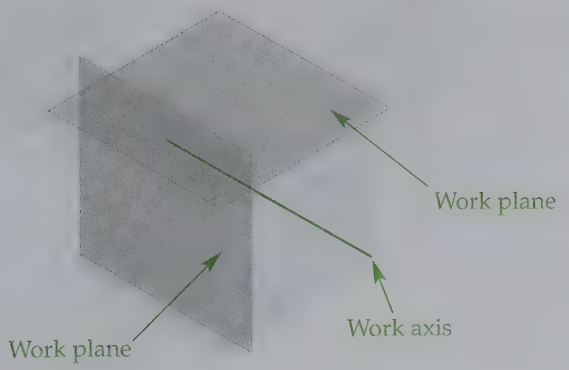


Figure 8-21.
Creating a work axis at the intersection of two work planes or feature faces.



- **Using a revolved or curved feature.** This option places a work axis through the middle of a revolved or curved feature, such as an extruded circle or a fillet, and is accomplished by picking the feature. See Figure 8-22.
- **From a linear feature edge.** Use this option to place a work axis collinear to a selected linear edge. To create a work axis by referencing an edge, pick the edge where you want the work axis to be located. See Figure 8-23.
- **From a face or plane and a point.** This option allows you to define a work axis perpendicular to the selected face or work plane and through a specified point. To create a work axis by referencing a face or plane and a point, pick the face or plane, and then pick any point including a feature edge endpoint, intersection, midpoint, center point, and work point. Or, select the point, followed by the face. See Figure 8-24.
- **Using a planar edge and a face.** This option allows you to define a work axis at the edge of a selected face, parallel to a specified edge. To create a work axis by referencing a face and an edge, pick the face and then pick an edge. Or, select the edge, followed by the face. See Figure 8-25.
- **Using sketch geometry.** This option allows you to use existing 2D and 3D sketch geometry to add work axes. There are multiple ways to use sketches to define the location of work axes. For example, you can add a work axis by selecting a linear 2D or 3D sketch line, or by picking sketch points such as endpoints, midpoints, and center points. Another option is to select a sketch point in conjunction with any existing work point, feature face, plane, or axis.

Figure 8-22.
Creating a work axis by picking a revolved or cylindrical feature.

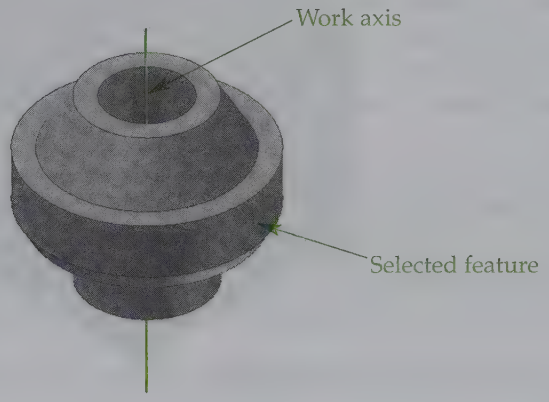
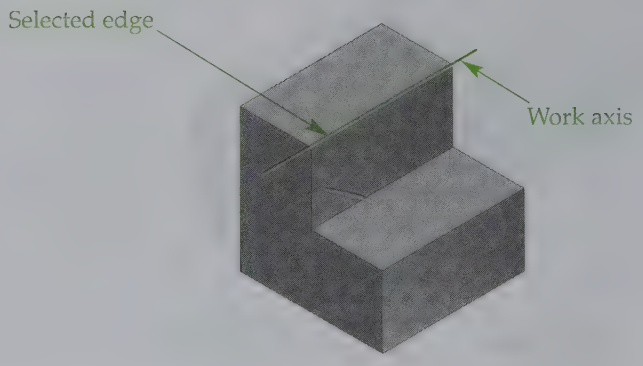


Figure 8-23.
Creating a work axis collinear to a linear edge.



Exercise 8-5 Complete the Exercise on the Student CD.

Exercise 8-6 Complete the Exercise on the Student CD.

Figure 8-24.
Creating a work axis by selecting a face or plane, and a point.

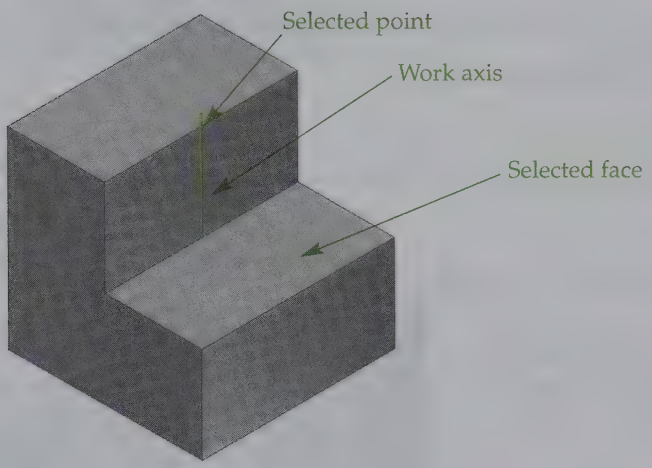
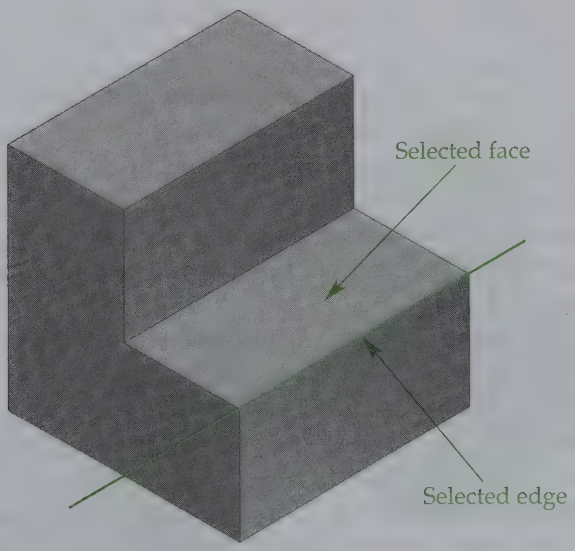


Figure 8-25.
Creating a work axis by selecting a face and an edge.

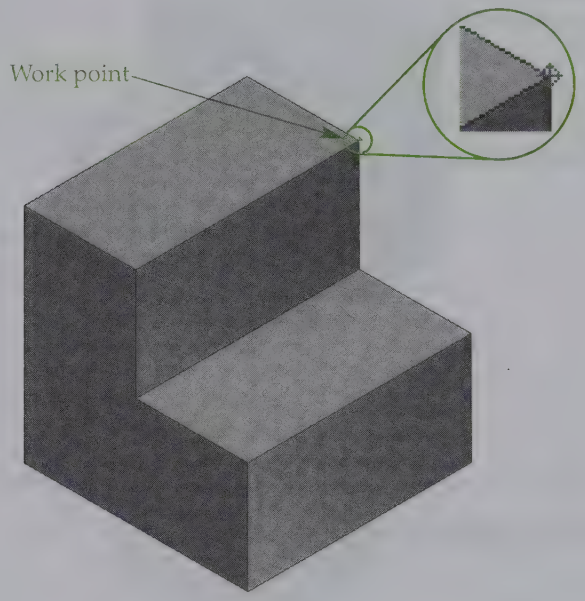


Using Work Points

A ***work point*** is a parametric reference point that can be located on any part feature or in three-dimensional space. Generally, you can use work points to create 3D sketch geometry, establish points at work plane and work axis intersections, place points at the intersection of three feature faces or planes, locate part vertices, such as edge endpoints, midpoints, or curve center points, help place a work axis or work plane, and create constraint points for assemblies.

Work points are displayed as highlighted work point icons in the graphics window and may be partially hidden by feature geometry. See Figure 8-26. If at any time you want to turn off the visibility of, or hide, a work point, right-click the work point in the browser and deselect the **Visibility** menu option, or right-click on the work point in the graphics window and deselect the **Visibility** menu option.

Figure 8-26.
A visible work point.

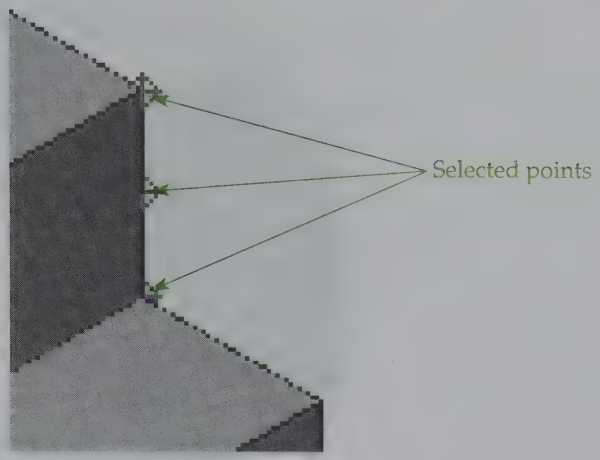


Work points are created using the **Work Point** tool that you access by picking the **Work Point** button on the **Part Features** panel bar; picking the **Work Point** button on the **Part Features** toolbar; picking the **Work Point** button on the **Work Point** toolbar; or pressing the [.] key on your keyboard.

When you access the **Work Point** tool, you place the work point by selecting existing reference geometry. Once you access the **Work Point** tool, use one of the following methods to create a work point:

- **Using default reference work features.** Although you can select the default work features located in the **Origin** folder of the browser to create a work point, you can accomplish the same task by referencing the center point. For example, if you pick the YZ plane, XZ plane, and XY plane, a work point identical to the default center point is created.
- **At existing feature vertices.** Use this option to generate a work point by referencing any available work point location, including a feature edge or curve endpoint, intersection, midpoint, and center point. To create a work point by referencing vertices, pick the point location. See Figure 8-27.

Figure 8-27.
Creating a work point by selecting a single point.



NOTE

You can also place work points at the vertices of sketch geometry.

- **Using three intersecting work planes or faces.** This option generates a work point at the intersection of three work planes or faces. To create a work point by referencing three work planes or faces, pick the work planes or faces where you want the work point to be located. See Figure 8-28.
- **From an edge or axis and a face, plane, or surface.** This option allows you to define a work point at the intersection of the selected face, plane, or surface and edge or axis. To create a work point by referencing a face, plane, or surface and an edge or axis, pick the face, plane, or surface, and then pick an edge or axis. Or, select the edge or axis, followed by the face, plane, or surface. See Figure 8-29.

Figure 8-28.
Creating a work point at the intersection of three work planes or feature faces.

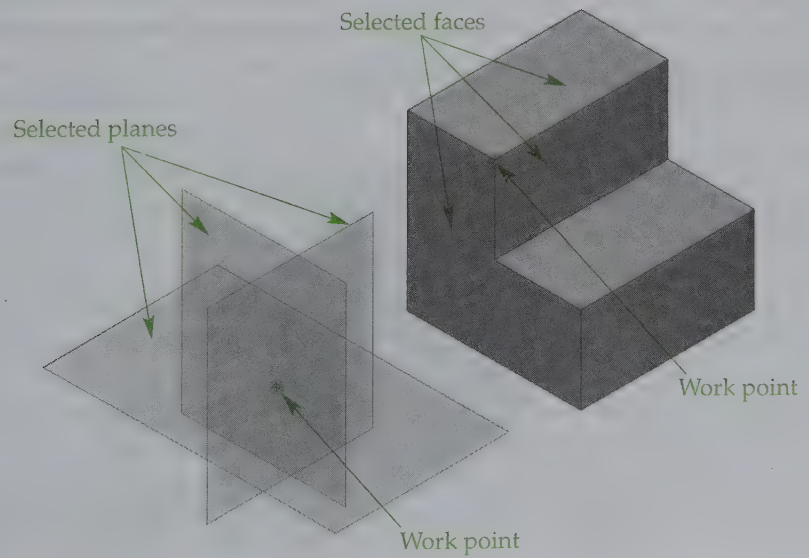
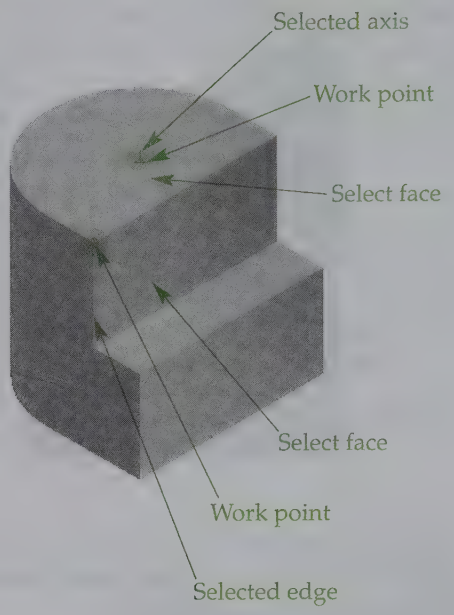


Figure 8-29.
Creating a work point by selecting a face or plane and an edge or axis.



- **From two feature edges or axes.** Use this option to define a work point at the intersection of two selected edges or axes. To create a work point by referencing edges or axes, pick the two edges or axes where you want to place the point. See Figure 8-30.
- **At the intersection of two projected coplanar edges.** Use this option to define a work point at the projected intersection of two selected coplanar feature edges. To create a projected work point by referencing edges, pick the two edges where you want to place the point. See Figure 8-31.
- **Using sketch geometry.** This option allows you to use existing 2D and 3D sketch geometry to add work points. There are a variety of ways to use sketches to define the location of work points. For example, you can define a work point by selecting two intersecting linear 2D or 3D sketch lines or by picking sketch points such as endpoints, midpoints, and center points. Another option is to select a sketch curve such as a circle, arc, spline, or ellipse in conjunction with any existing planar feature face or work plane.

Figure 8-30.
Creating a work point by selecting two edges or axes.

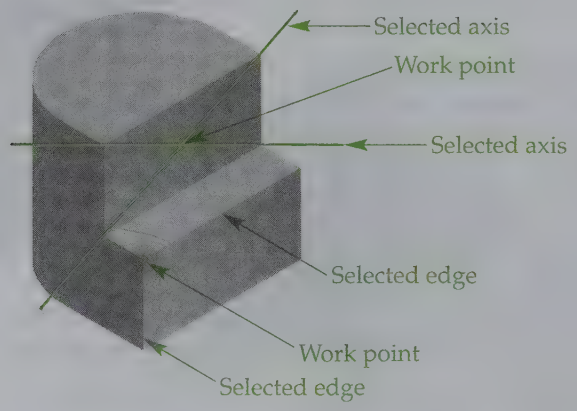
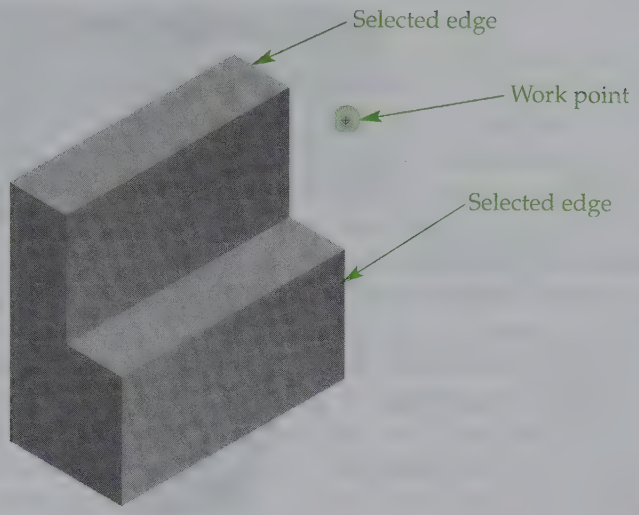


Figure 8-31.
Creating a projected work point by selecting two edges.



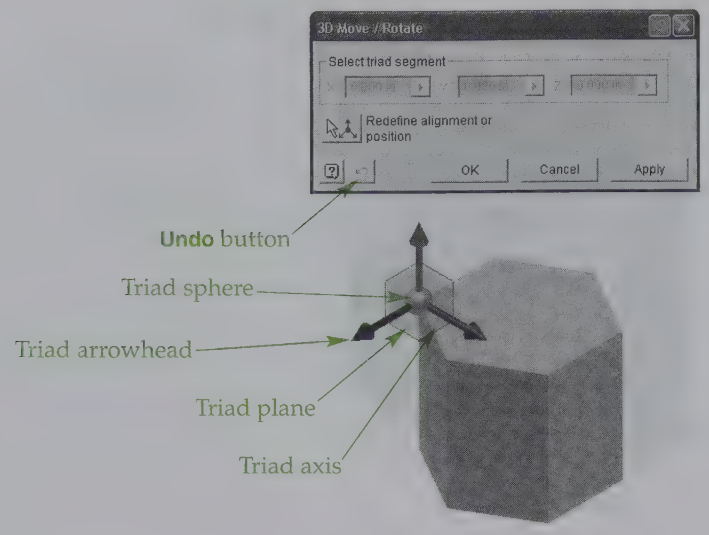
Adding Grounded Work Points

The work point flyout button and **Work Point** toolbar also contain the **Grounded Work Point** tool. A ***grounded work point*** is completely fixed to an X, Y, Z coordinate at which it is placed and is used to add a nonparametric work point to any location in three-dimensional space. Grounded work points cannot be fixed in reference to the model geometry on which they are placed, and therefore do not respond to model dimension or constraint changes. For example, if you place a grounded work point on the endpoint of a 5" line, and then change the length of the line to 6", the grounded work point remains at the initially placed X, Y, Z location, in this example, the 5" line endpoint.

There are two ways to create a grounded work point. The first option is to access the **Grounded Work Point** tool from the flyout button on the **Part Features** toolbar, **Part Features** panel bar, or **Work Point** toolbar, or by pressing the [;] key on your keyboard. Then define the initial location of the grounded work point by selecting any existing feature point, including an edge or curve endpoint, intersection, midpoint, center point, or work point. Once you specify the initial point, a triad and the **3D Move/Rotate** dialog box are displayed. See Figure 8-32. The grounded work point triad contains a sphere, three axes, three planes, and three arrowheads. You can select any of these individual triad items to manipulate the location and rotation of the work point. The selected triad item can also be used in conjunction with the precise input options in the **3D Move/Rotate** dialog box. Refer to Figure 8-33 as you read the following descriptions of the grounded work point operations:

- **Unrestricted move.** This option is available when you select the triad sphere. An unrestricted move allows you to move the grounded work point anywhere in three-dimensional space. You can manually move the point by holding down the left mouse button on the sphere and dragging the point to the desired location. Another option is to pick the sphere and then use the **X**, **Y**, and **Z** edit boxes in the **3D Move/Rotate** dialog box to identify a precise location for the move.
- **Planar move.** This option is available when you pick any of the three triad planes. A planar move allows you to move the grounded work point anywhere along the selected plane. You can manually move the point by holding down the left mouse button on the plane and dragging the point to the desired location.

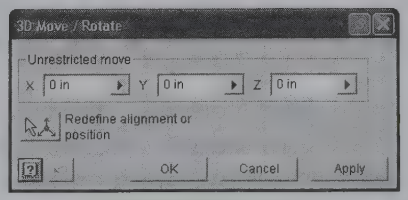
Figure 8-32.
A grounded work point triad and the **3D Move/Rotate** dialog box.



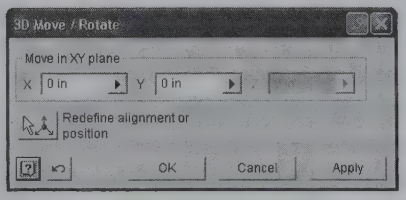
Another option is to use the corresponding edit boxes in the **3D Move/Rotate** dialog box to identify a precise location for the move. For example, if you pick the XY triad plane, the X and Y edit boxes are available for definition.

- **Axis move.** This option is available when you pick any of the three triad arrowheads. An axis move allows you to move the grounded work point anywhere along the selected axis. You can manually move the point by holding down the left mouse button on the arrowhead, and dragging the point to the desired location. Another option is to use the corresponding edit box in the **3D Move/Rotate** dialog box to identify a precise location for the move. For example, if you pick the Z triad axis, the Z edit box is available for definition.
- **Axis rotation.** This option is available when you pick any of the three triad axes. An axis rotation allows you to rotate the triad around the selected axis, if needed, allowing you to further position the grounded work point. You can manually drag the triad by holding down the left mouse button on the axis,

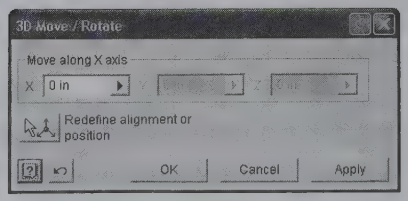
Figure 8-33.
The available edit boxes in the **3D Move/Rotate** dialog box changes based on the selected triad item.



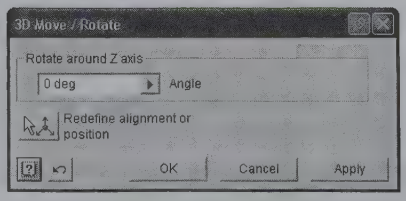
Unrestricted move



Move in XY plane



Move along X axis



Rotate around Z axis

and rotating the triad to the desired location. Another option is to use the corresponding options in the **3D Move/Rotate** dialog box to identify a precise rotation. Use the **Angle** edit box to enter or select the angle in degrees in which you want the axis to rotate.

If you want to locate a grounded work point at the initially selected point (0,0,0 in reference to the grounded work point triad), pick the **OK** button to generate the work point. However, if you move the work point or rotate the triad, you can either pick the **OK** button to generate the work point, or pick the **Apply** button to submit the movement or rotation, and continue moving the work point and rotating the triad as needed. In addition, if you want to return to the previous location and undo a movement, pick the **Undo** button. You may also have the option of selecting the **Redefine alignment or position** button which is used to reposition the work point at a specified location. If the triad sphere is selected, you can relocate the grounded work point by picking any available point in the model. If the triad axis is selected, you can align the selected axis with any available edge, axis, or sketched line, by picking the desired edge, axis, or sketch line. Finally, if a triad plane is selected, you can relocate the plane by picking any available face or work plane. Again, once you are satisfied with the location of the work point, pick the **OK** button.

The second option for creating a grounded work point is to place an ungrounded work point using the **Work Point** tool and any of the techniques previously described. Then right-click on the ungrounded work point and pick the **Ground** pop-up menu option. When you transform a work point into a grounded work point, the **3D Move/Rotate** dialog box does not automatically open. If you want to reposition the grounded point, you must right-click on the grounded work point and pick the **3D Move/Rotate** pop-up menu option.

NOTE



When you redefine the location of a grounded work point by right-clicking the grounded work point and selecting the **Redefine Feature** pop-up menu option, the grounded work point reverts to an ungrounded work point.



Complete the Exercise on the Student CD.



Complete the Exercise on the Student CD.



Complete the Exercise on the Student CD.

Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. Briefly describe *work features* and their use.
2. When can work features be created, and when are most work features created?
3. When you access the **Work Plane** tool or any other work feature, no dialog box opens. What do you do instead?
4. What does it mean when it is said that work planes appear *semitransparent* in the graphics window?
5. Identify the easiest way to create work features without using existing geometry.
6. What is the status of the panel bar mode when a feature is present and you are in the part feature work environment?
7. What is a *work plane*?
8. What does it mean when it is said that a work plane is *infinite*?
9. Work planes appear semitransparent in the graphics window. How do you turn off the visibility of, or hide, a work plane?
10. How do you move a work plane?
11. How do you change the displayed size of a work plane?
12. Moving or resizing the work plane display does not change the specified location, constraints, or area of the work feature. How do you physically relocate the work plane in space?
13. Describe a *work axis*.
14. What does it mean when it is said that a work axis is *infinite*?
15. Identify at least three uses for work axes.
16. Describe a *work point*.
17. Identify at least three uses for work points.
18. What is a *grounded work point*?
19. Explain the difference between a grounded and an ungrounded work point.
20. Give an example of a grounded work point.

Problems

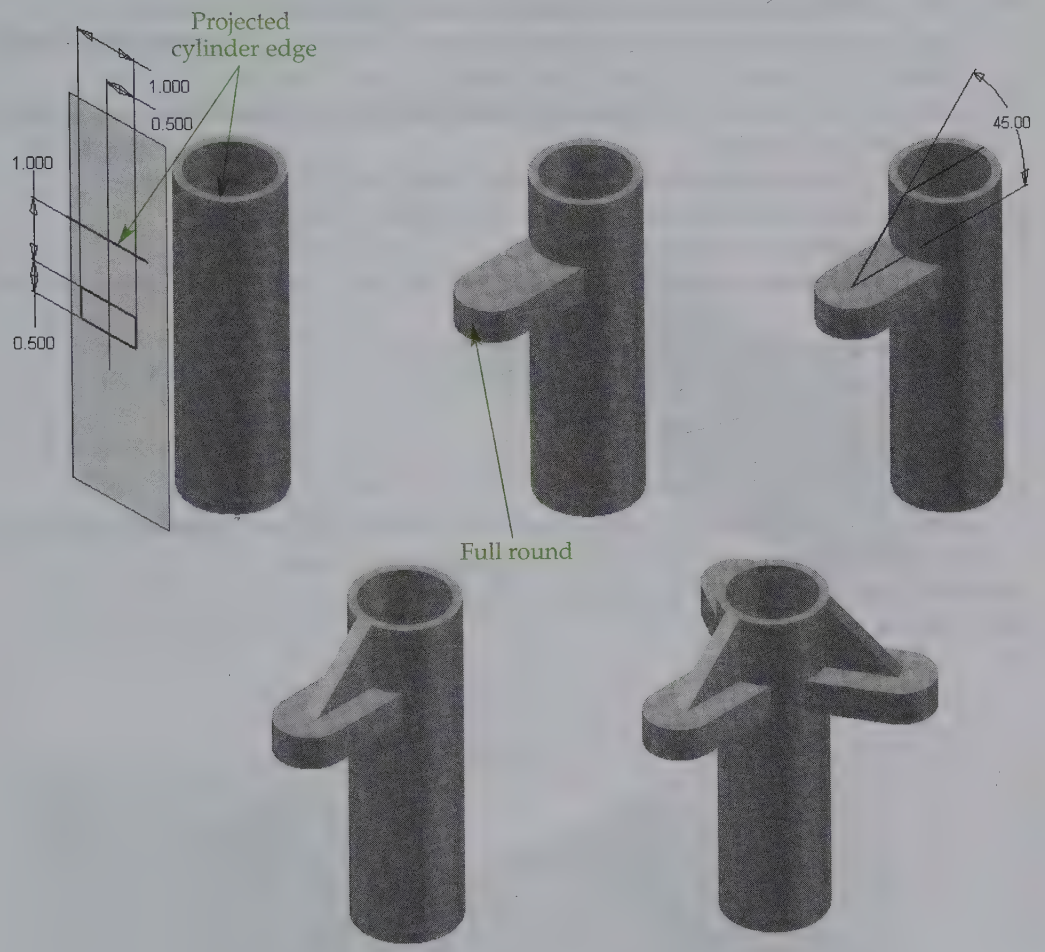
Instructions:

- Follow the specific instructions provided for each problem.

1. **Name:** Hub
   Units: Inch
   Template: PART-in.ipt
   Material: Default
   Color: Default
   Specific Instructions:
   - Open P7-7.ipt and save a copy as P8-1.ipt.
   - Close P7-7.ipt without saving and open P8-1.ipt.
   - Offset a work plane from the XY plane, 2.25".
   - Open a sketch on the work plane, and sketch the geometry shown.
   - Extrude the sketch in a negative direction, using the **To Next** termination option.
   - Add the round shown using the **Full Round Fillet** option (the work plane's visibility has been turned off for clarity).
   - Open a sketch on the YZ plane, and sketch the geometry shown.
   - Use the sketch to create a .25" thick rib, as shown.

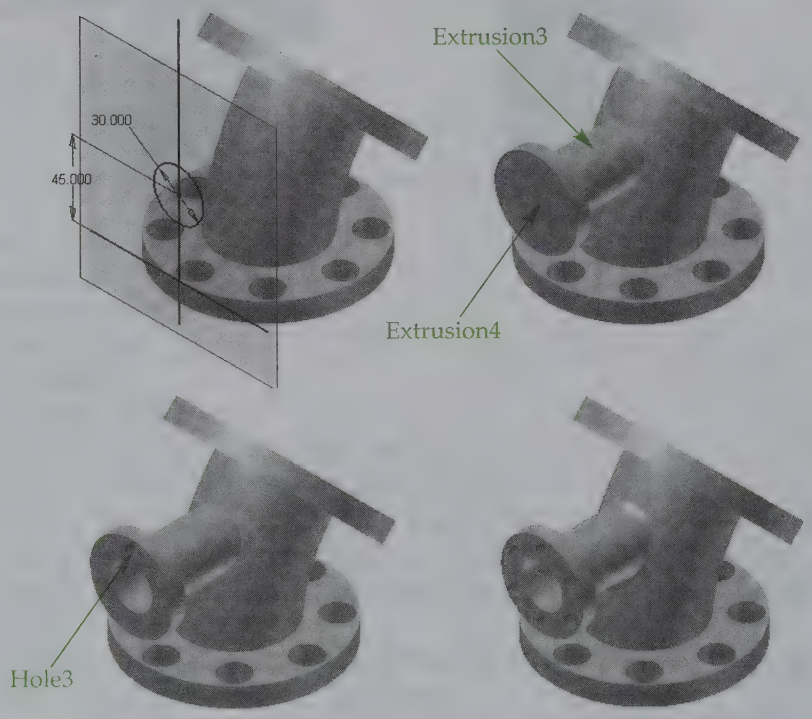
- Use the **Circular Pattern** tool to pattern the extrusion, using the following information:
  - **Rotation Axis:** Y axis
  - **Count:** 3
  - **Positioning Method:** Fitted
  - **Angle:** 360°

The final part should look like the part shown.
Save As: P8-1.ipt.



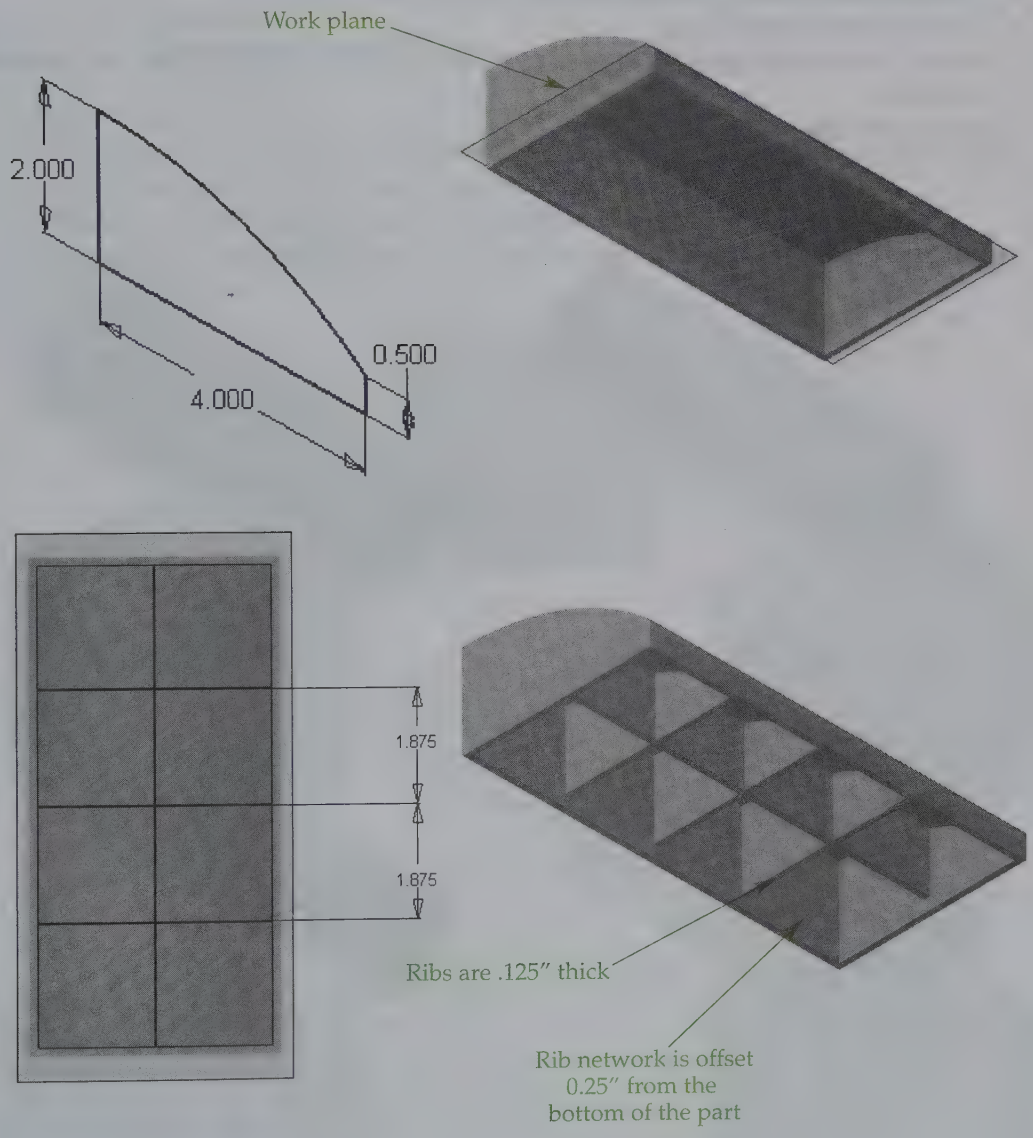
2. **Name:** 45° Elbow
Units: Metric
Template: PART-mm.ipt
Material: Steel, Mild
Color: As Material
Specific Instructions:
- Open P7-9.ipt and save a copy as P8-2.ipt.
- Close P7-9.ipt without saving and open P8-2.ipt.
- Create a work plane by selecting the XY plane and Extrusion1, as shown.
- Open a sketch on the work plane, and sketch the geometry shown.
- Extrude the sketch in a positive direction, using the **To Next** termination option.
- Create a ∅50 mm circle on face of Extrusion3, and extrude 8 mm in a positive direction, as shown.
- Add a ∅25 mm hole terminating at the inside face of the 45° elbow.
- Add the specified hole, 19 mm from the center of Extrusion4.
- Create a work axis by selecting Hole3.
- Use the **Circular Pattern** tool to pattern the extrusion, using the following information:
  - **Rotation Axis:** Work Axis1 in the browser
  - **Count:** 6
  - **Positioning Method:** Fitted
  - **Angle:** 360°
- Add a 5 mm fillet as shown.
- Open the **Properties** dialog box for the model and enter 45 DEGREE ELBOW in the **Title** edit box, and P8-2 in the **Part Number** edit box.

The final part should look like the part shown.
Save As: P8-2.ipt.



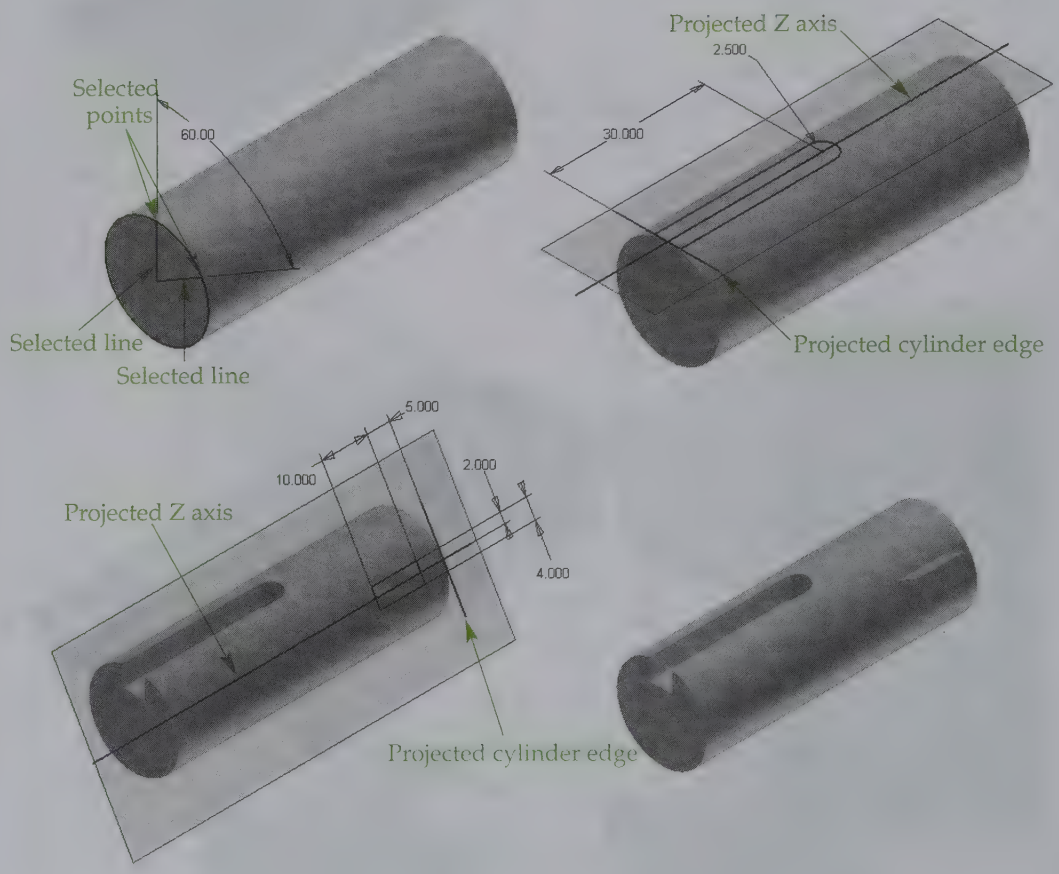
3. **Name:** Base
Units: Inch
Template: PART-in.ipt
Material: ABS Plastic
Color: Grey (Dark)
Specific Instructions:
- Begin a new inch part file.
- Create the sketch shown on the XY plane.
- Extrude the sketch 8″ midplane.
- Shell the extrusion .125″, removing the bottom lace as shown.
- Offset a work plane –.25″ from the bottom face as shown.
- Open a sketch on the work plane, and sketch the lines shown.
- Use the **Rib** tool to create the rib network shown.

Save As: P8-3.ipt.



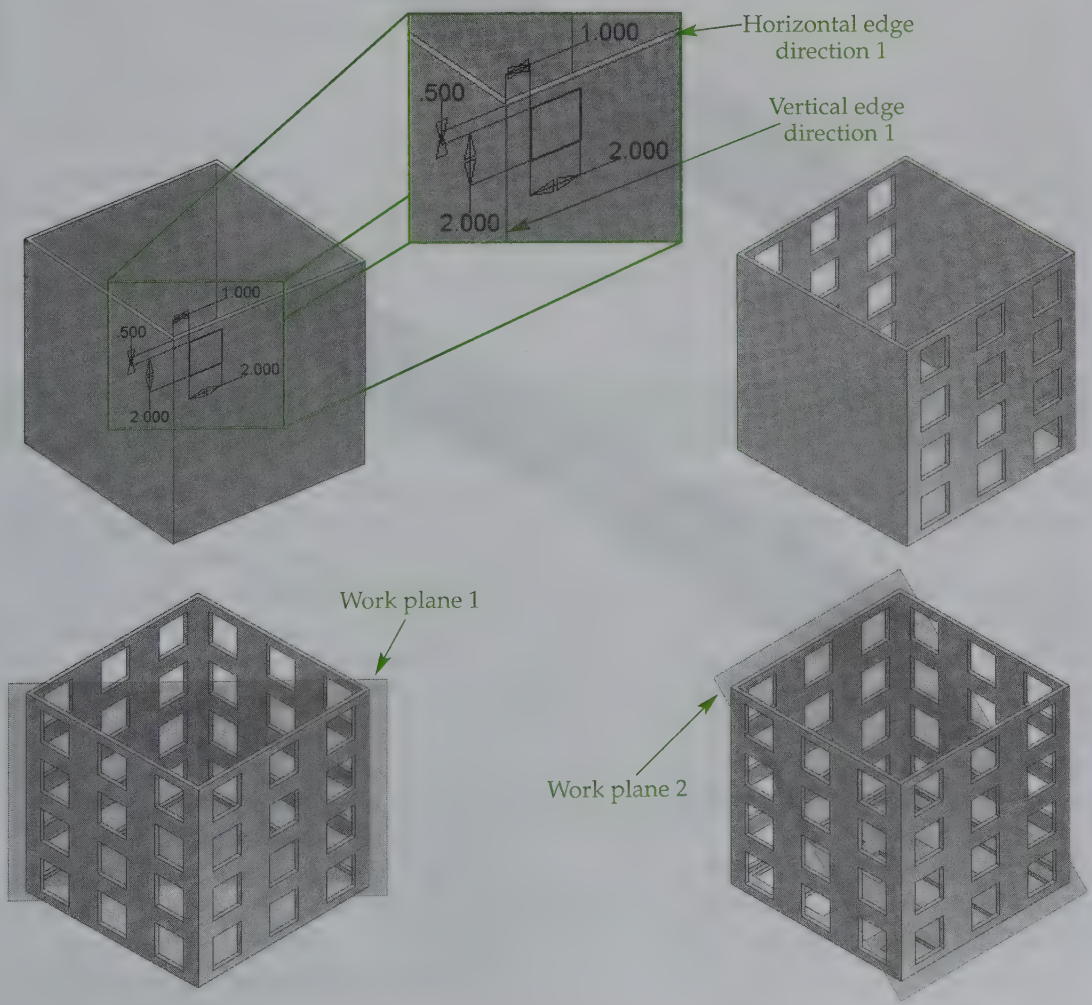
4. **Name:** Shaft
Units: Metric
Template: PART-mm.ipt
Material: Default
Color: Default
Specific Instructions:
- Begin a new metric part file.
- Sketch a ∅20 mm circle on the XY plane.
- Extrude the sketch 60 mm midplane.
- Sketch the geometry shown, on the left face of the cylinder.
- Share the sketch.
- Create work plane1 by picking the specified line and point.
- Sketch the geometry shown on the work plane.
- Cut-extrude the sketch 5 mm.
- Create work plane2 by picking the specified line and point.
- Sketch the geometry shown on the work plane.
- Extrude the sketch 5 mm.

The final part should look like the part shown with all work features, and Sketch2 hidden.
Save As: P8-4.ipt.



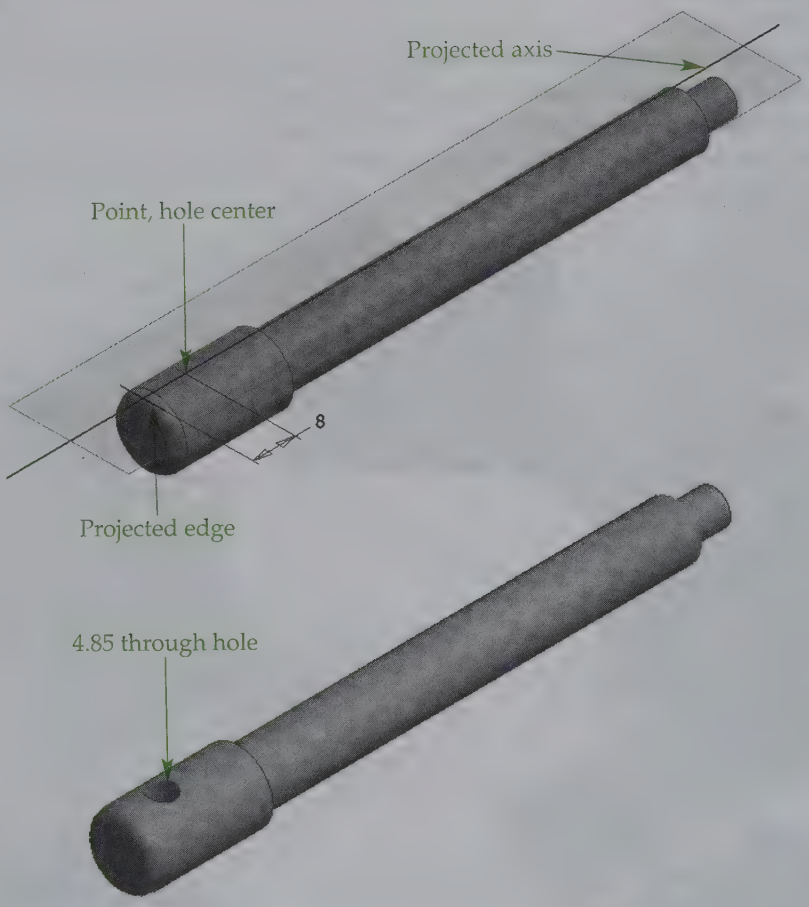
5. **Name:** Box
Units: Inch
Template: PART-in.ipt
Material: ABS Plastic
Color: As Material
Specific Instructions:
   - Begin a new inch part file.
   - Sketch a 12″ × 12″ rectangle on the XY plane.
   - Extrude the sketch 12″ in a positive direction.
   - Shell the box .25″ inside, removing the top face.
   - Sketch the geometry shown, on one of the box faces.
   - Cut-extrude the sketch using the **All** termination option.
   - Use the Rectangular pattern tool to pattern **Extrusion2** as shown, using the following information:
     - **Direction 1:** Horizontal edge
     - **Count:** 3
     - **Spacing:** 4″
     - **Direction 2:** Vertical edge
     - **Count:** 4
     - **Spacing:** 3″
   - Create Work Plane1 and Work Plane2 as shown by selecting exterior edges.
   - Mirror Rectangular Pattern1, across Work Plane1.
   - Mirror Rectangular Pattern1, across Work Plane2.

   The final part should look like the part shown with all work features hidden.
   Save As: P8-5.ipt.



6. **Name:** C-Clamp Screw
Units: Metric
Template: PART-mm.ipt
Material: Steel, High Strength Low Alloy
Color: As Material
Specific Instructions:
   - Open P6-3.ipt and save a copy as P8-6.ipt.
   - Close P6-3.ipt without saving and open P8-6.ipt.
   - Create a work plane tangent to the screw head, by selecting the tangent face and the appropriate default plane, as shown.
   - Open a sketch on the work plane, and sketch the geometry shown.
   - Place a 4.85 mm through hole at the sketched Point, Hole Center.
   - Open the **Properties** dialog box for the model and enter C-CLAMP SCREW in the **Title** edit box, SCREW in the **Part Number** edit box, and C-CLAMP SCREW in the **Description** edit box.

   The final part should look like the part shown.
   Save As: P8-6.ipt.





CHAPTER 9

Complex Features

Learning Objectives

After completing this chapter, you will be able to do the following:

- Create lofts
- Create sweeps
- Develop three-dimensional sketches
- Create split features
- Emboss and engrave
- Use additional surface tools and techniques
- Create decals

Models can require complex features that tools; such as extrude, revolve, or fillet; cannot create, or at least cannot create in one step, using one tool. For this reason, Inventor provides additional feature tools to help you create a variety of parts for specific applications. These additional tools allow you to generate lofts, sweeps, splits, embossments, decals, and other surface developments.

Creating Lofts

A ***loft*** references two or more profiles, located on different planes, and blends the profiles together to form a feature. Lofts are very similar to extrusions but allow you to create features that are geometrically more complex. See **Figure 9-1**.

Before you can create a loft, you must create two or more sections. ***Sections*** are sketches and existing feature faces that are used to develop loft features. Loft sections can be independent of existing geometry, as shown in **Figure 9-1**, or can reference a feature face, allowing you to loft from existing features. See **Figure 9-2**. For many applications, you can select an existing feature face as a loft section without creating a sketch of referenced edges. Face sections are identified as Edges. An alternative is to add sketches using the **Autoproject edges during curve creation** and the **Autoproject edges for sketch creation and edit** tools.

Typically, additional sketches used for lofting are created on work planes. For example, the second sketch used to create the lofted feature in **Figure 9-1** was placed on a work plane offset from the default XY plane. Another example is the circle sketched between the existing extrusions shown in **Figure 9-2**, which was placed on a work plane offset from one of the extrusion faces.

Figure 9-1.
This is a feature created by lofting the sketch of a circle and the offset sketch of a triangle.

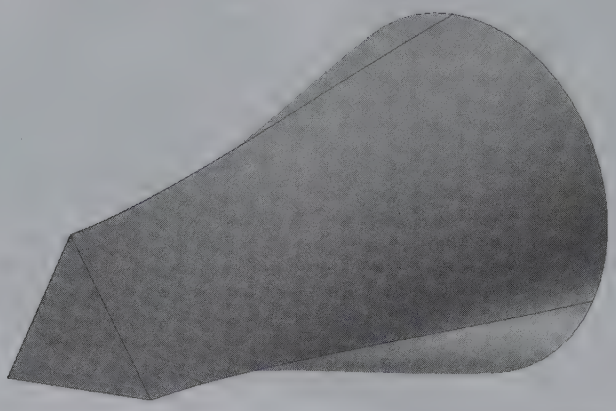
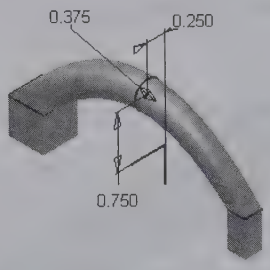
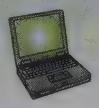


Figure 9-2.
A handle created by lofting two existing feature faces, and an additional offset sketch of a circle.



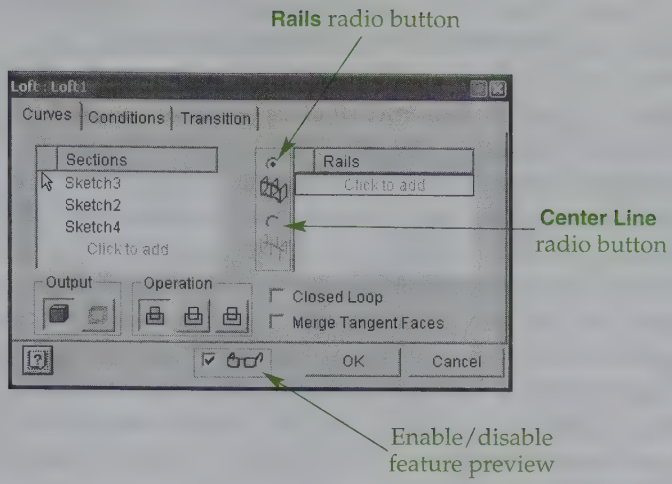
PROFESSIONAL TIP



When developing loft sketches, section sketches must be closed loop unless you want to create a *surface* loft; there is no limit to the number of sketch sections, or profiles, used to create a loft feature; and loft sections can be sketched at any angle and anywhere in space from another section, they do not have to be parallel.

Loft features are created using the **Loft** tool. Access the **Loft** tool to display the **Loft** dialog box by picking the **Loft** button on the **Part Features** panel bar; picking the **Loft** button on the **Part Features** toolbar; or pressing [Shift]+[L]. See Figure 9-3. To create a loft feature, you can first define the output by selecting either the **Solid** or the **Surface** button from the **Output** area. Picking the **Solid** button allows you to create a solid loft, while picking the **Surface** button allows you to create a volumeless surface loft. If you do not pick one of these buttons initially, the output is automatically specified depending on whether you pick open or closed loop sections. Once you define the output type, pick the **Click to add** button in the **Sections** list box and select sections to generate the loft. For simple lofts, such as those that contain only two sections, it is not critical which section you select first. However, for more complicated lofts, you must select sections in a logical, sequential order. Notice how selected sections are identified in the **Sections** list.

Figure 9-3.
The **Loft** dialog box, **Curves** tab.



PROFESSIONAL TIP



For clarity, you may want to pick loft sketches from the browser.

NOTE



A preview arrow identifying the direction of the loft is shown when you select the sections. If more than one possible sketch profile is available on a section sketch plane, you may have to pick the sketch plane and then pick the desired sketch profile.

Once you select all sections required to create the loft, pick the operation-type button from the **Operation** area. For simple base features, only the **Join** button is available. However, if other features are present in the part model, all the operation type buttons are offered, and you can choose to join, cut, or intersect the loft, depending on the application.

NOTE



The loft operation-type buttons function exactly the same as they do for other feature tools, such as **Extrude**.

At this point, many loft features can be formed by picking the **OK** button. However, if you are creating a more complex loft that contains several sections and you want to define additional shape and appearance characteristics, there are a variety of shape control options available. The following information describes each of the additional loft shape controls:

- **Closed Loop.** When you select this check box, located in the **Curves** tab of the **Loft** dialog box, a closed loop control is placed on a loft that contains three or more sections. This operation fixes the first and last sketch sections together, and is not available if you use rails.

- **Merge Tangent Faces.** In some cases abrupt edges are created between loft profiles, resulting in an unacceptable shape. If your design requires a smooth loft, you may be able to remove the abrupt edges and create the desired shape by selecting the **Merge Tangent Faces** check box.
- **Rails.** A *rail* is a 2D or 3D sketched curve that is used in conjunction with sections to manipulate and further control the loft shape. Figure 9-4 shows a loft created without a rail and the same loft created with a 2D sketched rail. A typical loft without a rail blends sections together from one section to another, along the straightest path. When a rail is added, sections are blended together from one profile to another, along the rail, creating an entirely different shape. A rail can be either a 2D or 3D sketched curve and must intersect a point on each loft section. You can create as many rails as needed to establish the desired loft shape. The following steps represent a typical approach to creating a 2D rail:

1. Add a work plane at the location where you want to sketch the rail.
2. If needed, place work points on each section to help define the location where the rail intersects each section. You do not need to add work points if you plan to use existing sketch curve points.
3. Open a sketch on the work plane you created during step 1.
4. Project sketched section points, such as curve endpoints, or the previously created work points, on the sketch plane to define the location where the rail intersects each section.
5. Use sketch curve tools to create the rail. Be sure the rail intersects the projected point on each of the sketched sections. See Figure 9-5.
6. To create a loft feature with a rail, select the **Rails** radio button. Then, pick the **Click to add** button in the **Rail** list box and select the sketched rail. Once the rail or rails are selected, you will notice the specified sketches are listed in the **Rails** list.

PROFESSIONAL TIP



Rails should be as smooth and uncomplicated as possible. You may not be able to generate a loft that contains very abstract rails.

Figure 9-4.
These models were created from the same profiles to illustrate the impact of adding a rail.

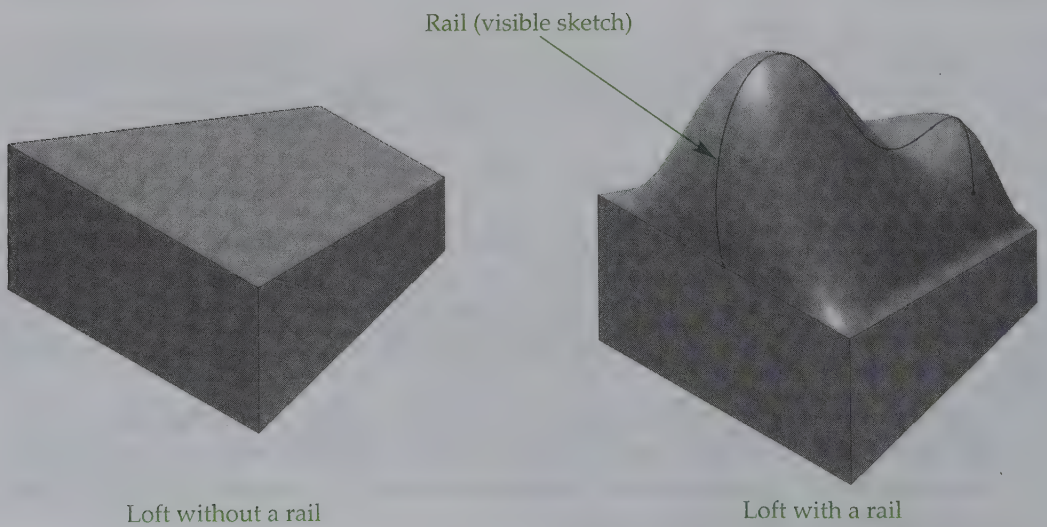
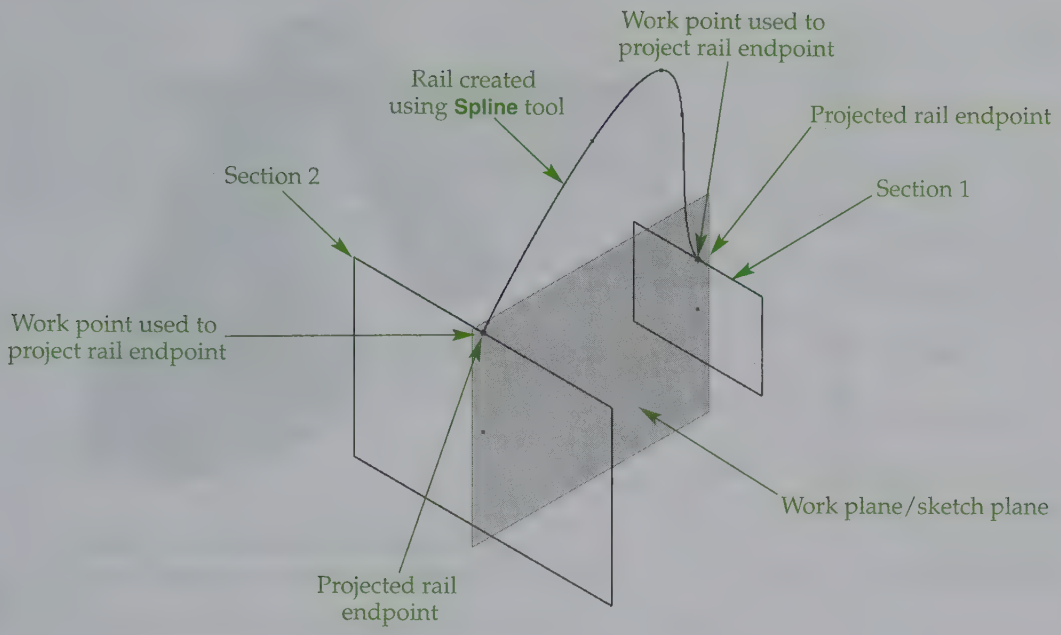
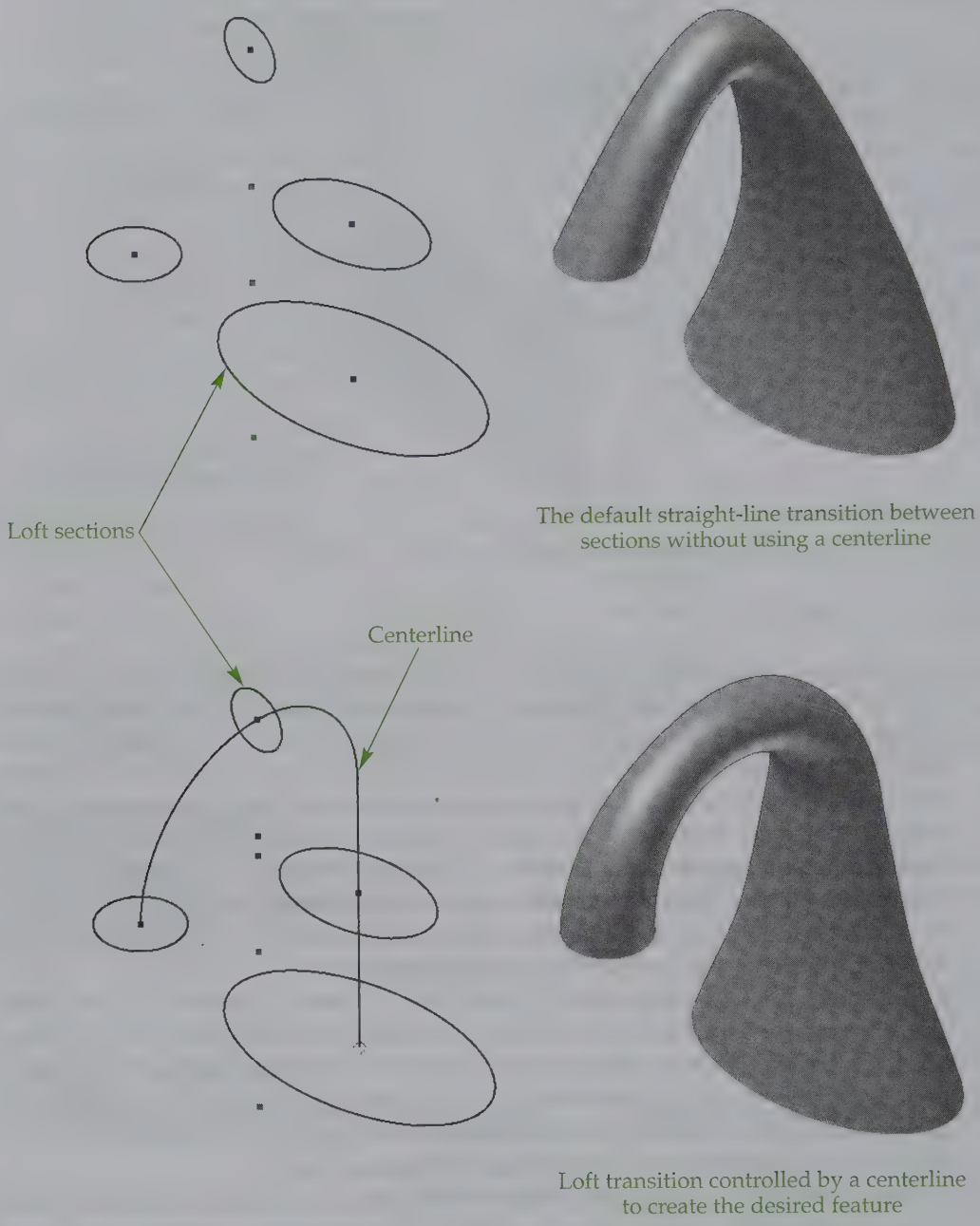


Figure 9-5.
Creating a loft rail.



- **Center line**. A ***center line*** is a rail that acts as a path for blending sections along and symmetrically around the ***center line*** sketch. Figure 9-6 shows a loft created without a ***center line*** and the same loft created with a 2D sketched ***center line***. Center lines are sketched just like rails. Typically a center line should be constrained to the center of sections. However, unlike rails, center lines do not need to intersect sections. Once you create sections and a center line, begin creating a loft feature using the center line by selecting the **Center line** radio button. Next, pick the **Click to add** button in the **Center line** list box and select the sketched center line. Once the center line is selected, you will notice the specified sketches are listed in the **Center line** list.
- **Conditions.** This tab, shown in Figure 9-7, is used to adjust the boundary characteristics of start and end sections, some rails, and sections that are points. No matter how many sections the loft contains, only the start and end sketch profiles are listed in the **Sections** list box. You can adjust the conditions by picking the desired sketch in the **Sections** list box and then selecting one of the following options from the **Conditions** drop-down list:
  - **Free Condition.** This option is selected by default and does not allow you to adjust the boundary angle or weight. A free condition applies a 90° angle and a weight of 0.
  - **Direction Condition.** This button is available only when the selected section is a 2D sketch profile. When you pick the **Direction Condition** button, the **Weight** edit box becomes available for definition. Then, if you enter a weight greater than 0, the **Angle** edit box becomes available. The **Angle** edit box allows you to specify the angle between the faces that the loft creates and the selected sketch profiles. See Figure 9-8.
  - **Tangent Condition.** This option is available only when the selected section is an existing feature face, was sketched on an existing feature face by projecting the face edges, or if the selected section sketch is adjoining a lateral surface or body. Selecting the **Tangent Condition** option places a tangent constraint between the section and the planar face from which the section was created. When you pick the **Tangent Condition** button, the **Weight** edit

Figure 9-6.
Using a loft center line to control the shape of a loft.



box becomes available for definition. The **Weight** edit box allows you to specify a unitless loft weight, which further defines the appearance of the loft. The desired weight is relative to the size of the loft feature and can range from 0 to infinite. However, weights that are too high may result in an undesirable loft shape. When you enter a small value for the weight, a more direct transition occurs between sketch profiles. See Figure 9-9. When you enter a larger number, a more indirect, meandering transition occurs between sketch profiles, depending on the specified weight.

- **Smooth (G2) Condition.** This option is available only when the selected section is an existing feature face, was sketched on an existing feature face by projecting the face edges, or if the selected section sketch is adjoining a lateral surface or body. Selecting the **Smooth (G2) Condition** option places

Figure 9-7.
The icons shown in the **Loft** dialog box, **Conditions** tab indicate the condition's type.

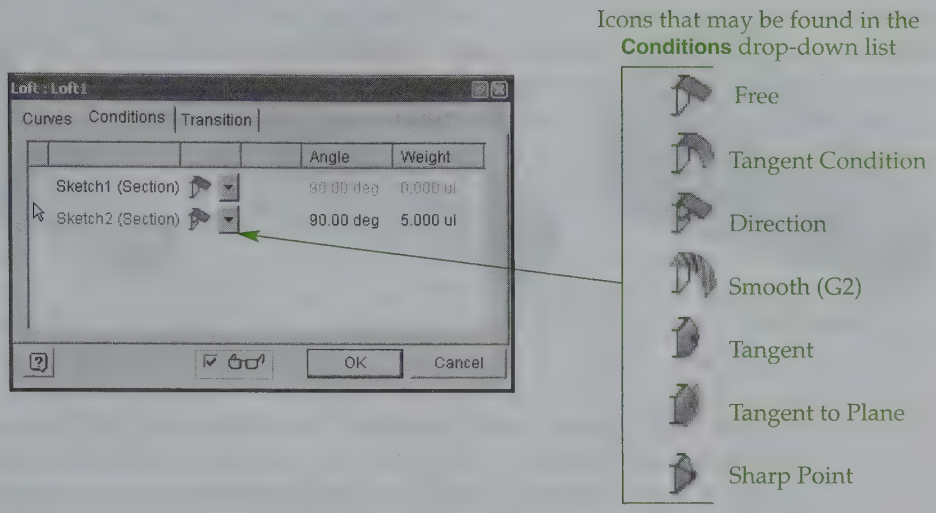
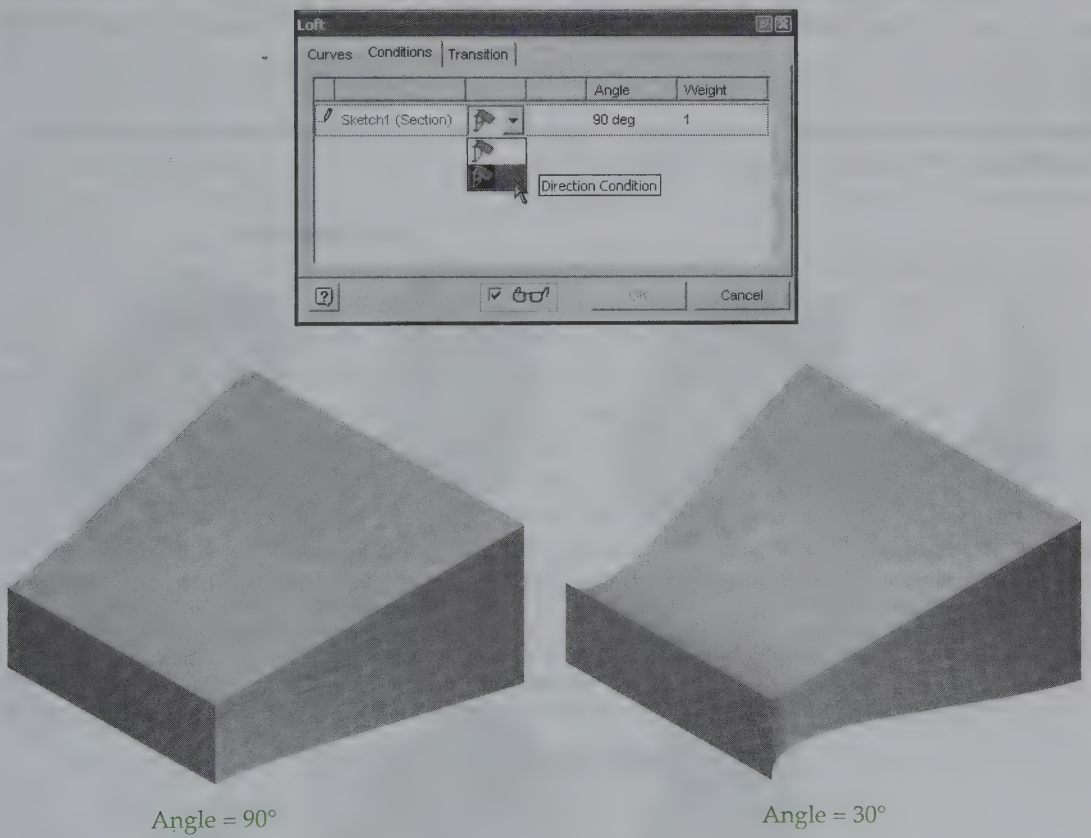
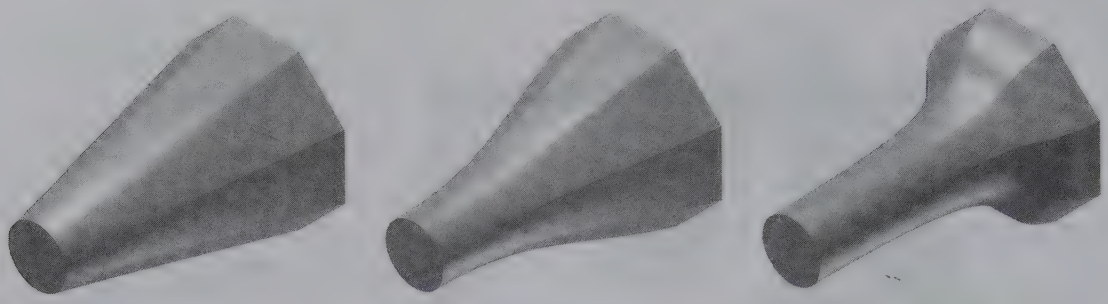


Figure 9-8.
An example of a loft with the default 90° angle and a 30° angle.



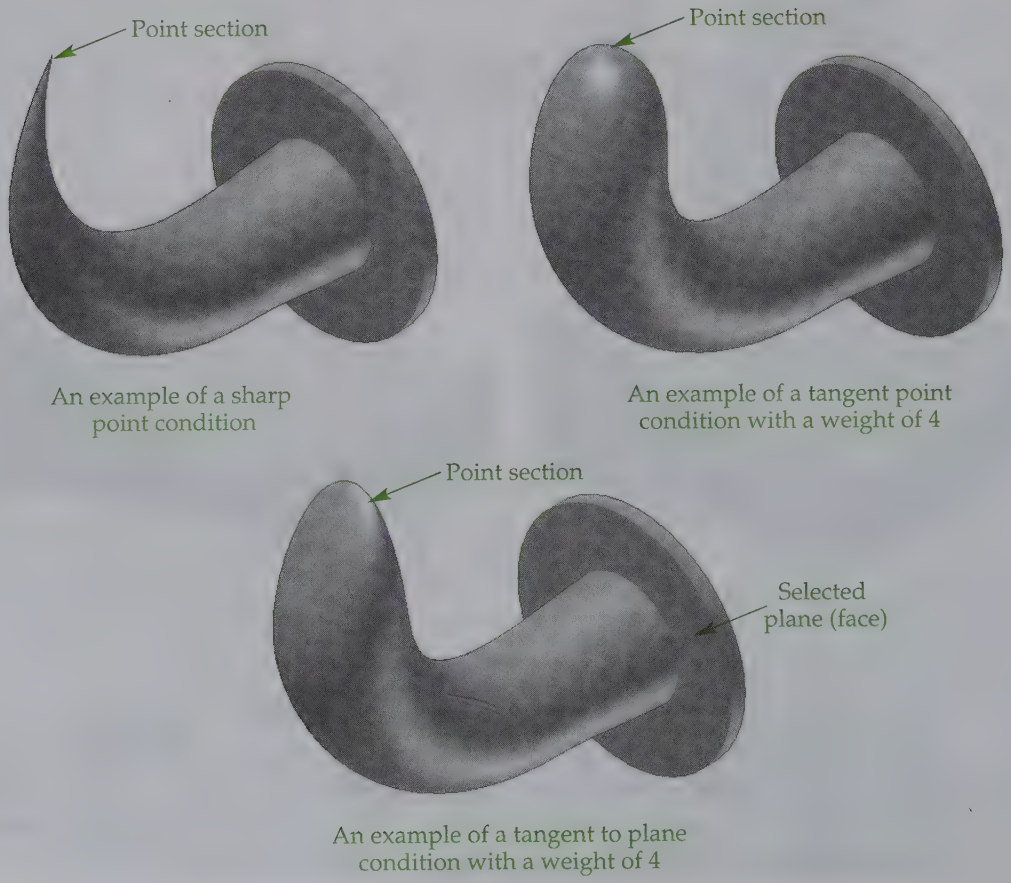
a smooth (G2) constraint between the section and the planar face from which the section was created. When you pick the **Smooth (G2) Condition** button, the **Weight** edit box becomes available for definition. The **Weight** edit box allows you to specify a unitless loft weight, which further defines the appearance of the loft, much like the tangent condition.

Figure 9-9.
These models were identical until the loft weight was changed.



- **Sharp Point.** This option is the default condition applied to a section that is a point. A sharp point condition creates a direct loft from a section to a point, without boundary condition. Use the **Sharp Point** option to create a pointed loft, such as the cone shown in **Figure 9-10.**
- **Tangent.** This option is available when the selected section is a point. Selecting the **Tangent** option creates a rounded point by applying a tangent constraint between the planar section and point. When you pick the **Tangent** button, the **Weight** edit box becomes available for definition. The **Weight** edit box allows you to specify a unitless loft weight. A higher weight value

Figure 9-10.
These lofts were created by using a point as one section. The point section condition can be adjusted according to the desired shape.



creates a rounder point section. Figure 9-10 shows an example of a lofted cone with a tangent condition weight of 4.

- **Tangent to Plane.** This option is available when the selected section is a point. Selecting the **Tangent to Plane** option creates a rounded, often flattened point based on the orientation of a selected plane or face. To create a **Tangent to Plane** condition, pick the **Tangent to Plane** button, followed by selecting a plane or face. The **Weight** edit box becomes available allowing you to specify a unitless loft weight. A higher weight value creates a rounder and often flatter point section. Figure 9-10 shows an example of a loft with a tangent to plane condition weight of 4.

- **Transition.** This tab helps you ensure that complex lofts are created correctly by mapping transition points. See Figure 9-11. By default, the **Automatic Mapping** check box is selected, which automatically defines the loft path transition, given the specified sections, rails or center line and shape conditions. However, when multiple sections or sections arranged in a more complicated fashion are used to create a loft, Inventor may not be able to clearly identify the correct loft transition. As a result, the automatic mapping function may select the incorrect transition points causing the loft to build an incorrect part. See Figure 9-12. Fortunately, point mapping usually takes care of this problem by allowing you to manually align sketch sections and correctly specify the loft path using selected points on each section.

To manually map points on each section, deselect the **Automatic Mapping** check box. As shown in Figure 9-13, when you deselect the **Automatic Mapping** checkbox, all the automatically defined transition path sets are listed in the **Point Set** list box and the automatically specified points used to map the loft are displayed in the **Map Point** list box in the sequence in which the sections were selected. Next to each map point sketch is a **Position** edit box that identifies the map point location relative to the line on which it is placed. A position of 0 defines one end of the line, while a position of 1 defines the opposite end of the line. For example, if you want to place a map point in the middle of the line, enter a position of .5.

There are a number of options for redefining automatically calculated map points. One option is to pick the point set you want to modify from the **Point Set** list box, which highlights the point set in the graphics window. You can then move your cursor over the map point you want to move, and without clicking on the point, drag it to the desired location and pick. Another option is to adjust specific map points by entering a new position for the point in the selected **Position** edit box. In some cases, if the loft is extremely twisted and numerous point sets are automatically created, as shown in Figure 9-12, it may be easier to remove all automatic point sets and create a new set, or remove and modify all but one or two point sets. Point sets can be removed by picking the desired point set and pressing the [Del] key. If you choose to remove all the automatic point sets

Figure 9-11.
The **Loft** dialog box with the **Transition** tab revealed and **Automatic Mapping** checked.

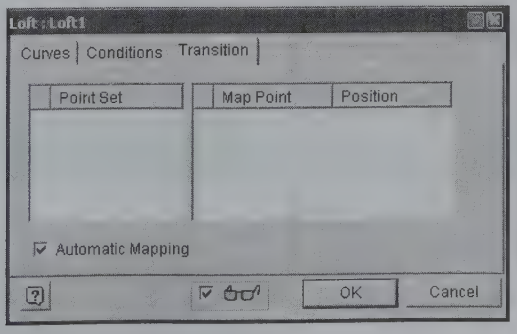
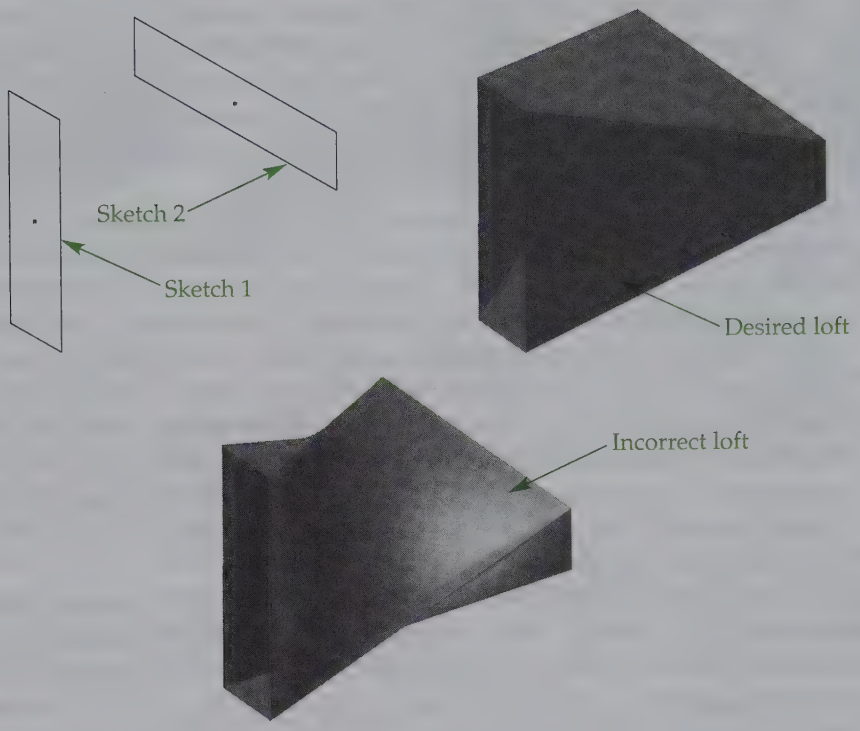


Figure 9-12.
The two sketches were selected to produce a simple transition from a vertical rectangle to a horizontal rectangle. Unfortunately, the automatic mapping created the incorrect twisting shape.



or just want to create an additional point set, pick the **Click to add** button in the **Point Set** list box. Then select a point on each of the sketch sections in sequential order. Usually, you can pick any available point on a sketch, but it is very important that you choose corresponding points on each sketch. See **Figure 9-14.**

Once you select all loft items and identify the required options in the **Loft** dialog box, pick the **OK** button to generate the loft feature.

Figure 9-13.
Deselect the **Automatic Mapping** check box to see the list of point sets and to change the position of any incorrect point sets.

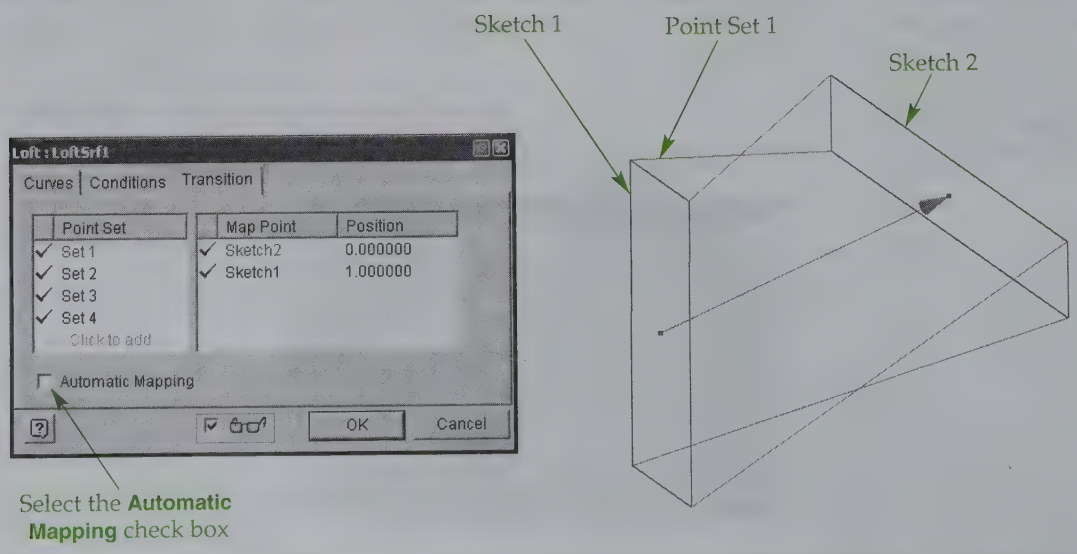
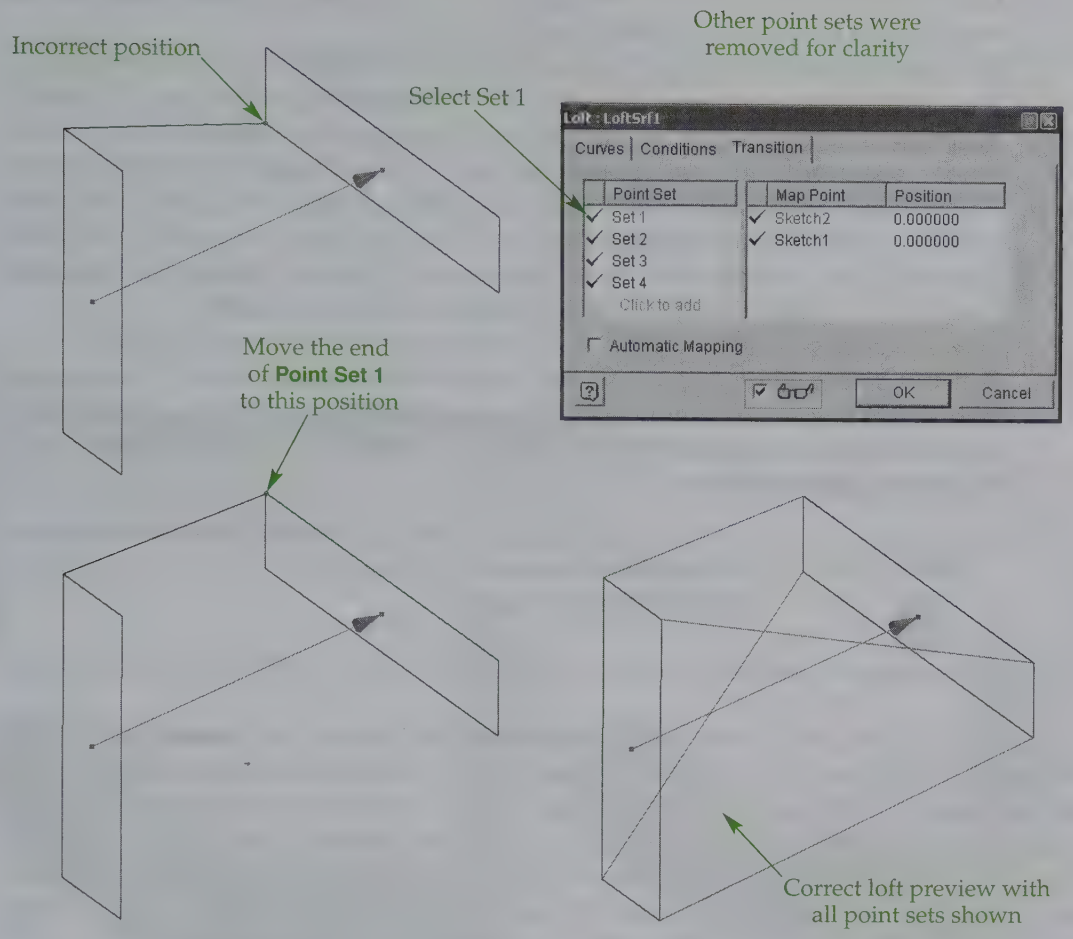


Figure 9-14.
Move the ends of each incorrect point set with manual point mapping to create the correct loft.



Exercise 9-1 Complete the Exercise on the Student CD.

Exercise 9-2 Complete the Exercise on the Student CD.

Exercise 9-3 Complete the Exercise on the Student CD.

Creating Basic Sweeps

A *sweep* is a feature that is created by referencing a sketch profile and a sketch path. The sketch profile follows, or is swept along, the sketch path. Sweeps are similar to extrusions but are geometrically more complex because they allow you to extrude a sketch along an irregular, nonlinear path. See Figure 9-15.

Before you can create a sweep feature, you must create two or more sketches. The following information is important for understanding how to create sweep sketches:

- One of the sketches must be the profile that you want to sweep, and another sketch must be the sweep path. See Figure 9-15.
- The profile and path must be located on intersecting planes. For example, if you create a profile on an XY plane, you can create the path on an XZ plane or YZ plane. Though the profile and path sketches must be created on intersecting planes, the profile and path sketch geometry do not need to be perpendicular.
- The profile does not have to intersect the path. See Figure 9-16. Still, it is better to have a profile that intersects the path, for dimensioning and constraining purposes, and to ensure the sweep is accurately made. Typically, a coincident constraint is used to establish an intersecting profile and path.
- Sweep sketches can be independent of existing geometry, as seen in Figure 9-16, or they can be placed on a planar feature face, allowing you to sweep on an existing feature. See Figure 9-17. If you want to use existing feature edges as a sweep profile or path, use the **Autoproject edges during curve creation** and the **Autoproject edges for sketch creation and edit** tools.

NOTE



Profile sketches must be closed loop unless you want to create a surface sweep.

Sweep features are created using the **Sweep** tool. Access the **Sweep** tool to open the **Sweep** dialog box by picking the **Sweep** button on the **Part Features** panel bar; picking the **Sweep** button on the **Part Features** toolbar; or pressing [Shift]+[S]. See Figure 9-18. To create a sweep feature, first select either the **Solid** output button to create a solid sweep feature, or select the **Surface** output button to create a volumeless surface sweep. Next, pick the **Profile** button if it is not currently active, and pick the sweep profile sketch. Then pick the **Path** button and select the sweep path sketch.

NOTE



When creating a simple sweep feature, the profile and path may be automatically selected when you access the **Sweep** tool. Once the profile and the path are selected, you can see a preview of the sweep.

Figure 9-15.
These sketches were used to create this simple sweep feature.

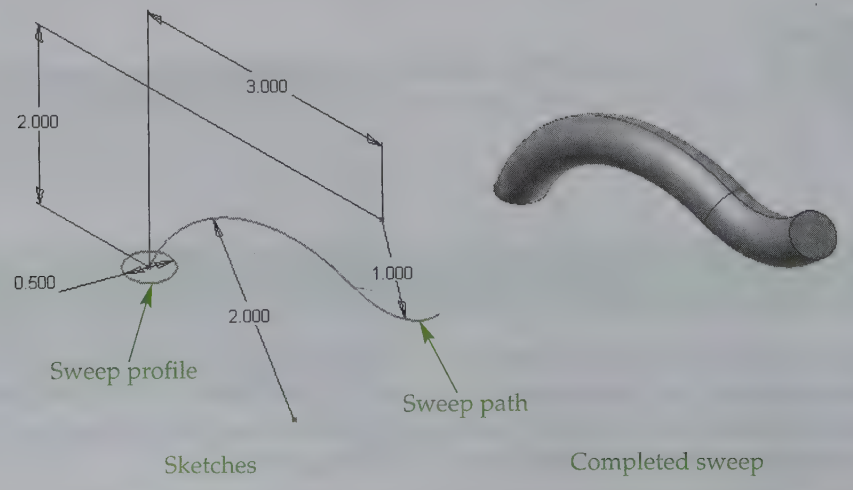


Figure 9-16.
A sweep can be constructed whether the path and profile intersect or not.

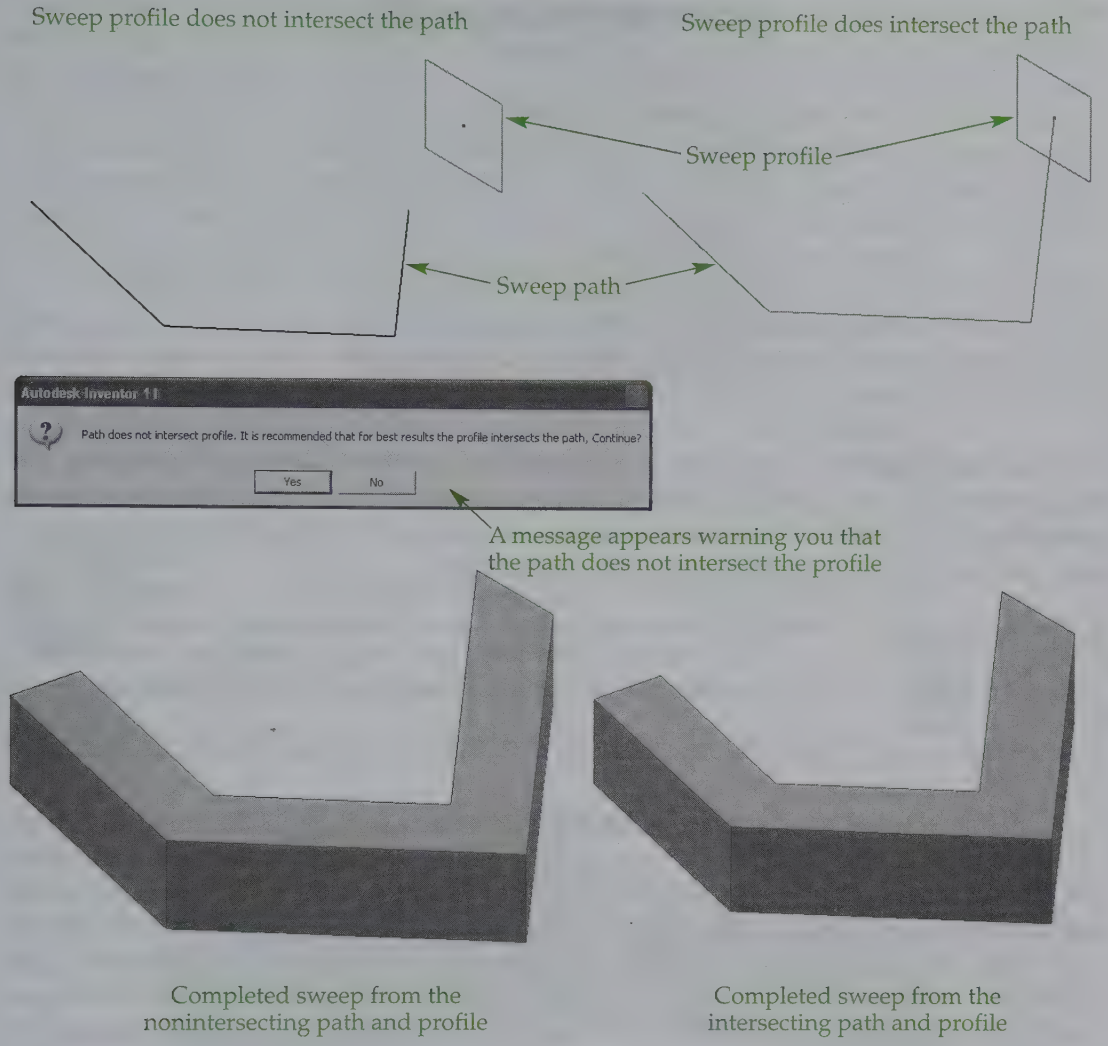


Figure 9-17.
Creating a sweep profile and path by referencing existing geometry. The profile is coincident to the corner endpoint of the extrusion, and the path is automatically projected from the top face.

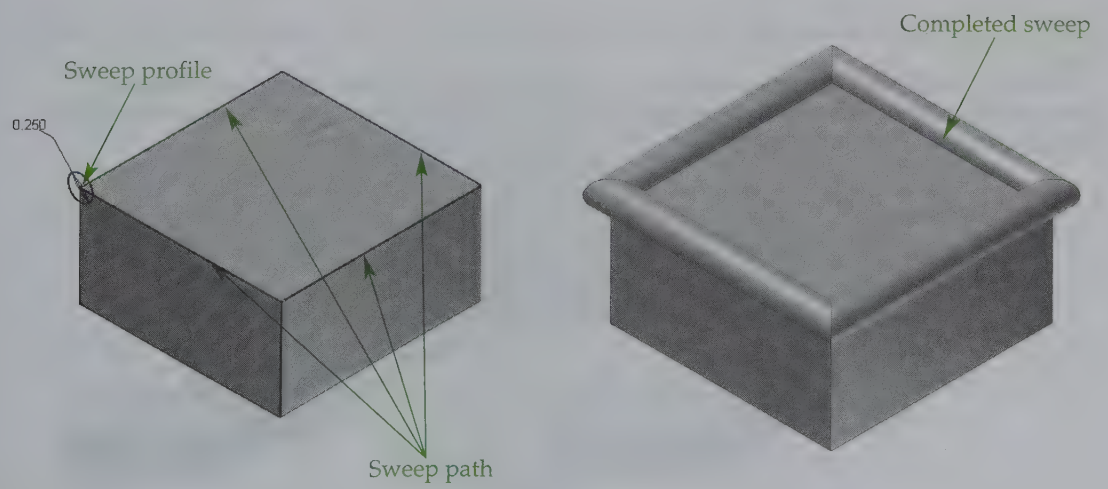
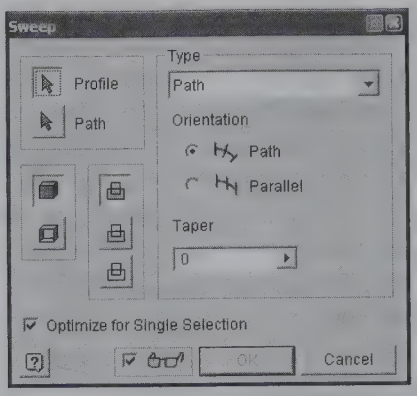


Figure 9-18.
The **Sweep** dialog box, **Shape** tab.



The next step in creating a sweep feature is to pick the desired operation type button. For base features, only the **Join** button is available. However, if other features are present in the model, all the operation type buttons are offered and you can choose to join, cut, or intersect, depending on the application.

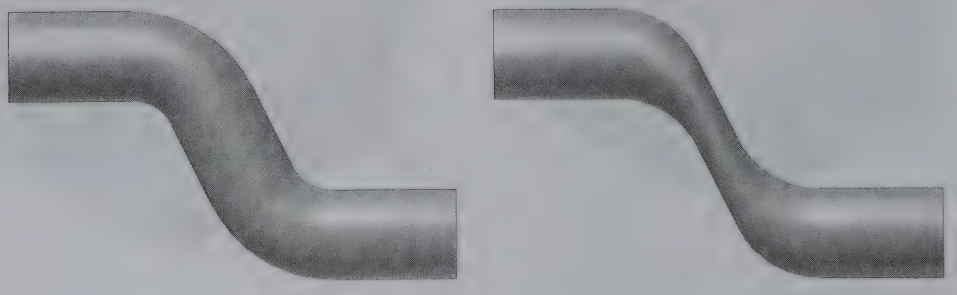
NOTE



The sweep operation type buttons function exactly the same as those in other feature tools, such as **Extrude**.

Once you select the output, sketch profile, sketch path, and the operation type, use the options in the **Type** area to further define the sweep. Select **Path** from the drop-down list to create a basic sweep feature based on a profile and a path. Choose the **Path** radio button to produce a sweep that maintains a perpendicular constraint between the profile and the path. Pick the **Parallel** radio button to create a sweep that becomes parallel to the profile when the path is not perpendicular to the profile. Figure 9-19 shows the difference between the **Path** and **Parallel** options. When using the **Path** option, you can specify a taper angle using the **Taper** edit box. See Figure 9-18. The taper angle defines the increasing or decreasing taper of the sweep profile as it follows the sweep path. The default angle of 0 applies no taper to the sweep resulting in a feature that is completely controlled by the sketch profile. A positive taper angle constantly increases

Figure 9-19.
The profile of a **Path** type of sweep feature created using the **Path** option maintains a perpendicular constraint between the profile and the path. The **Parallel** option of the **Path** type produces a different sweep, even with the same profile and path.



Path option (top view shown)

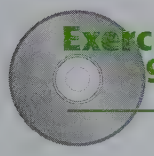
Parallel option (top view shown)

the size of the sweep profile. A negative taper angle constantly decreases the size of the sweep profile. See Figure 9-20.

Typical sweeps are created using the **Path** type, while the **Path & Guide Rail** and **Path & Guide Surface** types are available for specific applications. Choose **Path & Guide Rail** from the drop-down list to include a rail in the definition of a sweep. Sweep rails function very similarly to loft rails. To use a guide rail, pick the **Guide Rail** button, if it is not currently active, and select the 2D or 3D sketch guide rail. See Figure 9-21. Profile scaling controls the shape of a guide rail sweep. Pick the **X & Y** radio button to scale the sweep horizontally and vertically along the path; choose the **X** radio button to scale the sweep horizontally along the path; or select the **None** radio button to eliminate scaling. Figure 9-21 shows examples of each profile scaling option.

Pick **Path & Guide Surface** from the drop-down list to direct the form of a sweep along a surface. A guide surface is often required when a sweep profile follows a path along a complex surface. To use a guide surface, pick the **Guide Surface** button, if it is not currently active, and select the surface. Figure 9-22 shows an example of when a guide surface is required in order to produce the correct sweep shape.

Once you define all options in the **Sweep** dialog box, pick the **OK** button to generate the sweep feature.



Complete the Exercise on the Student CD.

Figure 9-20.
These models compare the effects of a positive and a negative taper angle.

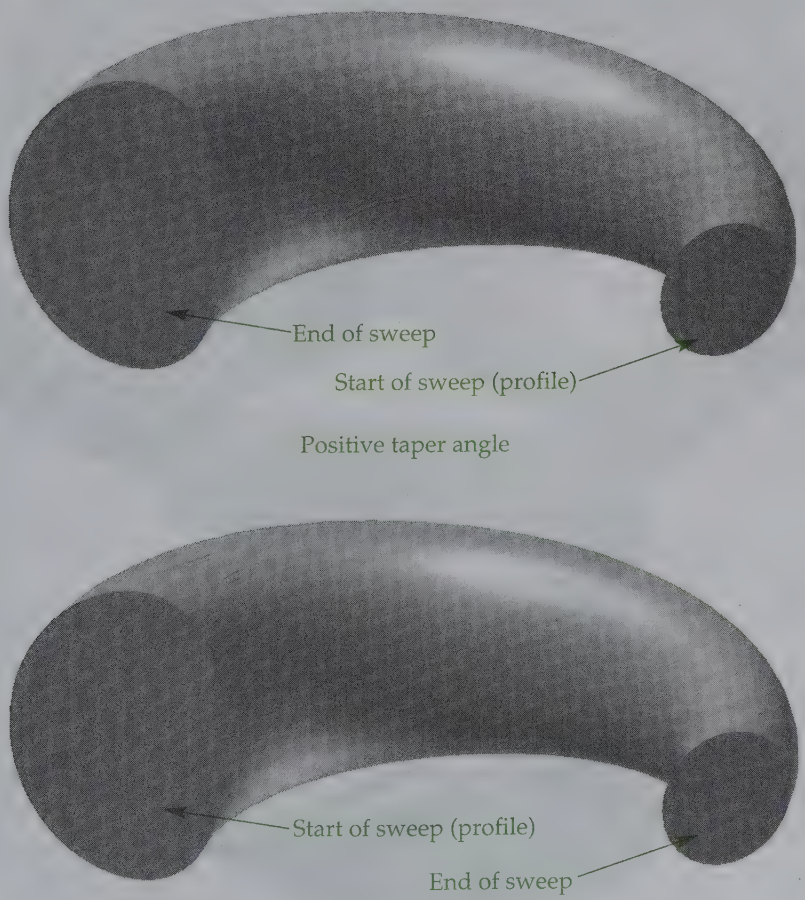


Figure 9-21.
Using a guide rail to produce a sweep feature.

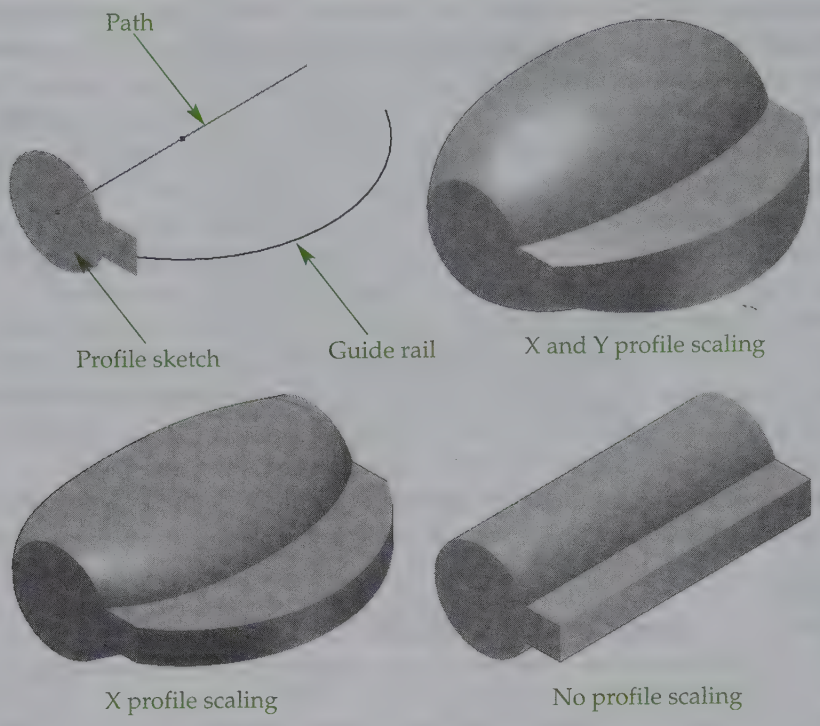
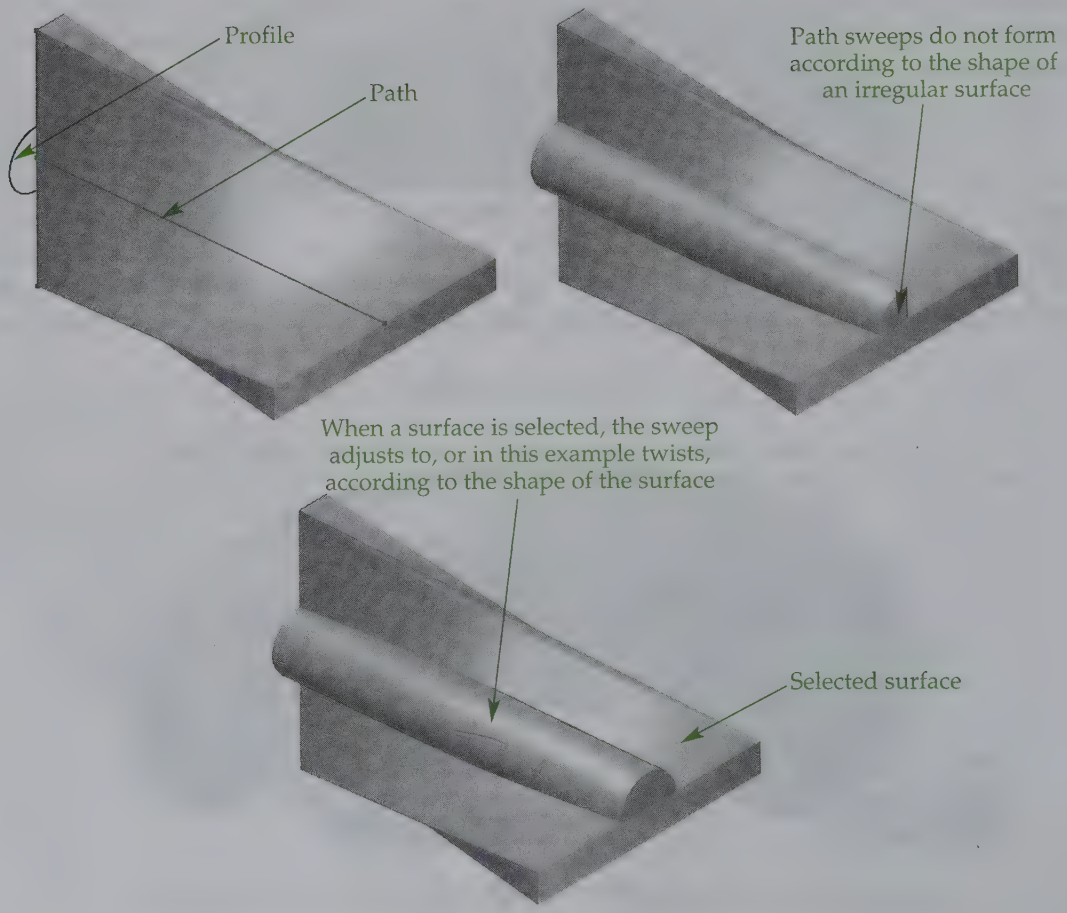


Figure 9-22.
Using a guide surface to direct the shape of a sweep.



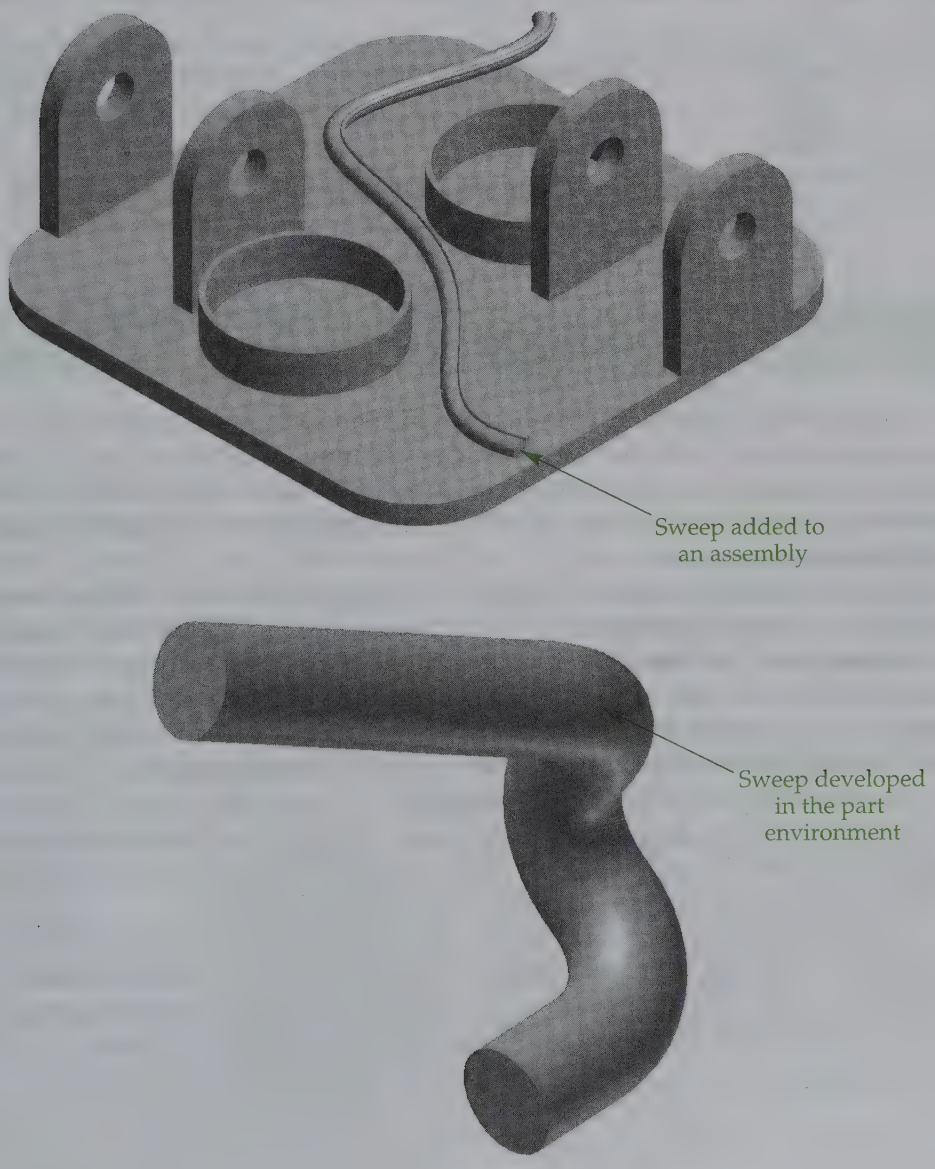
Creating Advanced Lofts and Sweeps

Advanced lofts and sweeps are created by referencing 3D sketches. An advanced loft uses 3D rails or a 3D center line to form a more complex feature. An advanced sweep follows a 3D path and can include a 3D rail and guide surface. A 3D sweep feature is used in various applications, such as to produce routed wires or pipes around existing features. 3D sweeps are often placed in assemblies but can be constructed in parts. See **Figure 9-23**.

NOTE

The process for creating a three-dimensional sweep feature is the same whether you add the sweep to an assembly or build it as a part.

Figure 9-23.
An example of a three-dimensional sweep feature added to an assembly and developed in the part environment.



3D loft and sweep features are very similar to 2D loft and sweep features. Except for using 3D sketches, the tools used to create advanced lofts and sweeps function exactly the same as those used to create basic lofts and sweeps. The main difference between a 2D and 3D loft or sweep is the use of 3D rails, a center line or paths, which are created using 3D sketch tools. Use the following techniques to sketch loft sections and 3D loft rails or a center line:

1. Open new 2D sketches and create loft sections.
2. Open a new 3D sketch and create a 3D sketch rail or center line.
3. Access the **Loft** tool and generate the loft feature using the 3D sketch rail or center line.

Use the following techniques to sketch a sweep profile and a 3D sweep path:

1. Open a new 2D sketch and create a sweep profile.
2. Open a new 3D sketch and create a 3D sketch path.
3. Access the **Sweep** tool and generate the sweep feature.

The process for sketching 2D loft sections and a 2D sweep profile is exactly the same as previously described. Once the sections or profile are complete, open a new 3D sketch. You can do this by right-clicking in the graphics window or the browser and selecting the **New 3D Sketch** menu option or by picking the **3D Sketch** flyout button on the **Inventor Standard** toolbar. When you open a new 3D sketch and are in 3D sketch mode, some specific 3D sketch environment and interface options are available.

NOTE



3D sketches are created in 3D space and are not placed on a single plane, or planar face, like a 2D sketch.

Using 3D Sketch Tools

You are already familiar with most of the common Inventor interface items using to create sketches. However, each work environment, including the 3D sketch environment, contains specific interface characteristics. Before you sketch 3D objects, you should understand the general 3D sketching items found in the **Document Settings** and **Options** dialog boxes.

The only functions specific to the 3D sketch environment in the **Document Settings** dialog box are the **3D Sketch** area of the **Sketch** tab and the **3D Snap Spacing** area of the **Modeling** tab. The **Sketch** tab is specific to the sketch environment and allows you to adjust the auto-bend radius for a 3D sketch in the **3D Sketch** area. When you create a 3D line, using the **3D Line** tool, you can choose to automatically place fillets and rounds at the intersecting corners of two lines by specifying a radius greater than 0 in the **Auto-Bend Radius** edit box. The **Modeling** tab is specific to the 3D sketch environment and allows you to set default 3D snap settings. The **3D Snap Spacing** edit box allows you to set the snap spacing for 3D sketching in inches, while the **Angle Snap** edit box is used to set the snap spacing for angles in degrees.

The only function specific to the 3D sketch environment in the **Options** dialog box is the **Auto-Bend with 3D Line Creation** check box located in the **3D Sketch** area of the **Sketch** tab. When creating lines in a 3D sketch environment, selecting this check box creates radii, or corner bends at the corners of lines without you having to place corner bends manually.

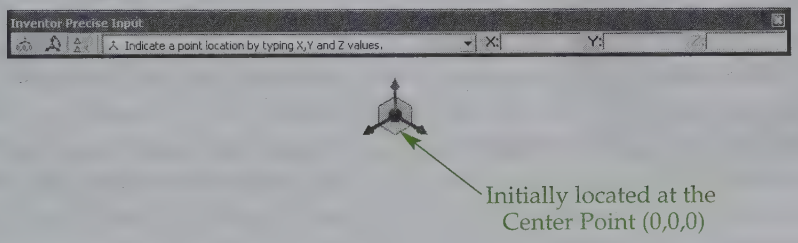
Sketching 3D Lines and Splines

There are three primary options for sketching 3D lines and splines. One method is to pick points in space, and then add dimensions and constraints. Another option is to use the **Inventor Precise Input** toolbar to define the location of X, Y, and Z coordinates. The third technique is to select work points, existing feature edge endpoints, or sketch points to define the location of lines and splines. Often a combination of these three techniques is used, depending on the application.

To create a line or spline using the **Inventor Precise Input** toolbar, activate the **Line** or **Spline** tool. The **Line** tool is available by picking the **Line** button on the **3D Sketch** panel bar or **3D Sketch** toolbar; or when no other tools are active, right-clicking inside the graphics window and selecting the **Create 3D Line** menu option. Access the **Spline** tool by picking the **Spline** flyout button on the **3D Sketch** panel bar or **3D Sketch** toolbar.

When you select either the **Line** or **Spline** tool, the **Inventor Precise Input** toolbar automatically appears, and the triad is positioned at the center point. See **Figure 9-24**. You do not have to use the **Inventor Precise Input** toolbar or triad to sketch lines and splines; you can just pick locations in space. For example, the sweep shown in **Figure 9-25** is created using a 3D sketch path of three lines with endpoints at random locations. However, the **Inventor Precise Input** toolbar is effective for precisely positioning line and spline endpoints.

Figure 9-24.
The **Inventor Precise Input** toolbar and triad.



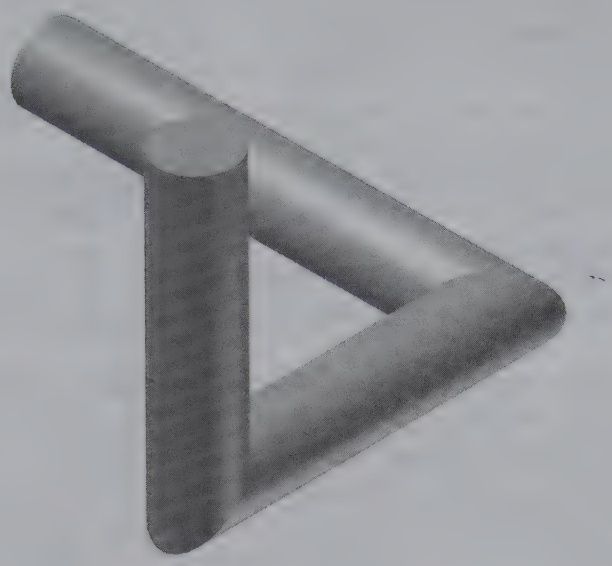
Using the **Inventor Precise Input** toolbar to locate line and spline endpoints involves entering values in the **X**, **Y**, and **Z** edit boxes. Type the value of the X coordinate in the **X** edit box, the Y coordinate in the **Y** edit box, and the Z coordinate in the **Z** edit box, and press [Enter] to create the point. Use this technique to continue creating lines and splines as needed. For example, to sketch a line that begins at the center point, and ends at coordinates 1, 3, 3, enter 0 in the **X**, **Y**, and **Z** edit boxes and press [Enter]. Next, type 1 in the **X** edit box, 3 in the **Y** edit box, and 3 in the **Z** edit box, followed by pressing [Enter]. When you are done, press [Esc] or right-click and pick the **Done** option.

NOTE



The **Precise Relative** and **Delta Input** buttons are automatically selected when you create 3D lines and splines. As a result, the first point of any new object is located at the temporary origin. You can select a triad plane to add a 0 to the X, Y, or Z edit box and define the location of a line or spline endpoint according to the selected plane.

Figure 9-25.
A sweep created using a 3D path.



To create a line or spline by referencing existing work points, feature edge endpoints, or sketch points, ensure that work points or endpoints are available, and pick the points in sequential order. See Figure 9-26. If you want to continue the line or spline, pick additional points. If not, press the [Esc] key or right-click and pick the **Done** option.

If the **Auto-Bend with 3D Line Creation** check box is activated, bends are automatically placed at line corners. The radius of the bend depends on the value specified in the **Auto-Bend Radius** edit box. An alternative method for activating or deactivating the **Auto-Bend with 3D Line Creation** tool is to access the **Line** tool, right-click, and select the **Auto-Bend** pop-up menu option. If the **Auto-Bend with 3D Line Creation** tool is not activated, no curve is placed at 3D line corners. If you want to add bends to unbent corners, access the **Bend** tool by picking the **Bend** button on the **3D Sketch** panel bar, or pick the **Bend** button on the **3D Sketch** toolbar.

Accessing the **Bend** tool opens the **3D Sketch Bend** dialog box, shown in Figure 9-27. To create a 3D sketch bend without using the **Auto-Bend with 3D Line Creation** option, first specify a bend radius in the edit box by typing a radius or picking a value from the list. Then select the **Equal** button if you want to create bends with equal radii, without adding additional dimensions. Next, select the intersecting corners of lines or the two lines that you want to bend.

NOTE

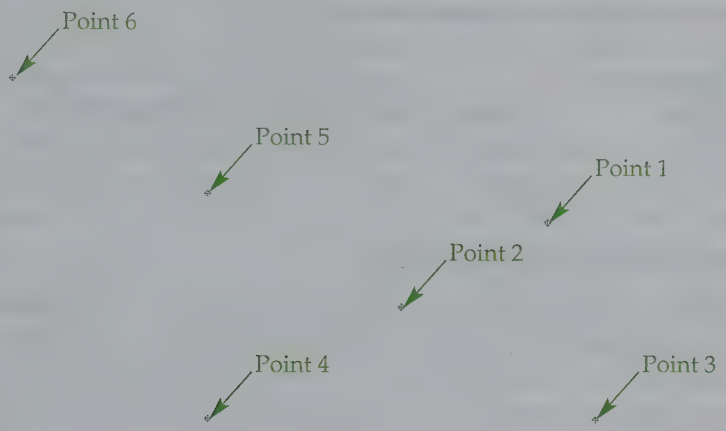


Splines are continuous lines with rounded corners. As a result, the line intersection bend options previously described do not apply to splines. However, all of the spline display options described earlier in this book are available when creating 3D splines.

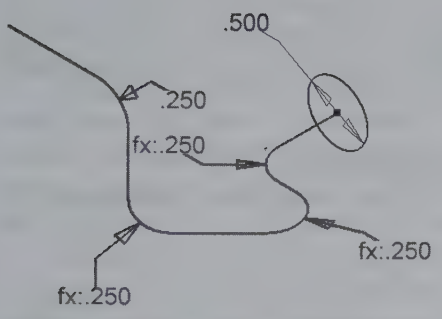
Referencing and Projecting Geometry

Any existing feature edge or 2D sketch shape can be used as a piece of, or the entire, 3D sketch. To use existing 2D sketch or model elements and convert them into 3D model geometry, access the **Include Geometry** tool by picking the **Include Geometry**

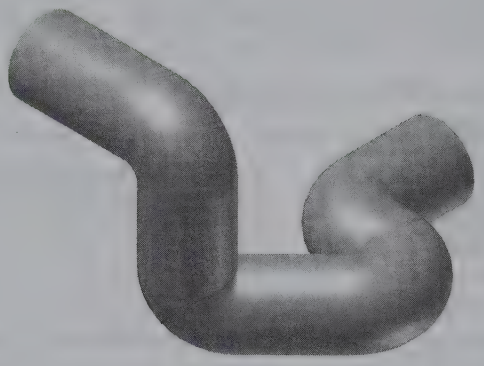
Figure 9-26.
These images illustrate the method of picking points to create a 3D sketch that will be used to make a 3D sweep feature.



Work points used to create 3D sketches

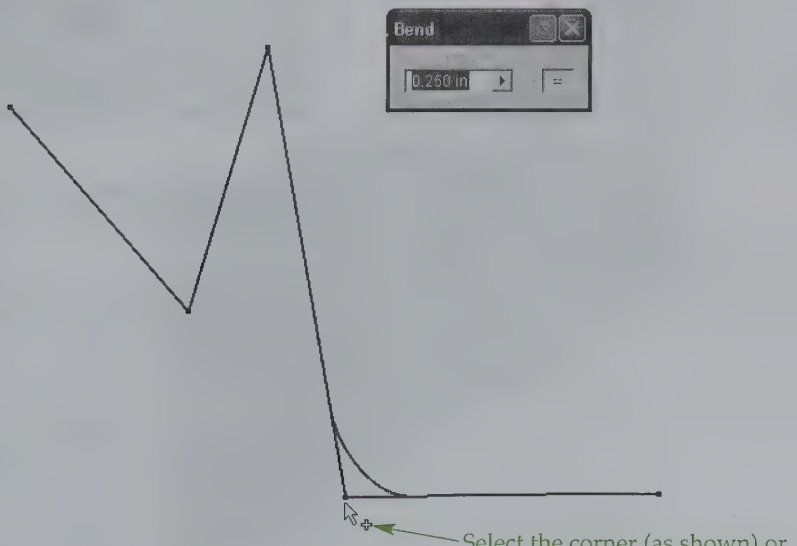


3D sketches



Completed sweep

Figure 9-27.
Creating a 3D sketch bend using the **3D Sketch Bend** dialog box.



button on the **3D Sketch** panel bar or picking the **Include Geometry** button on the **3D Sketch** toolbar. Next, pick the 2D sketch geometry and the model edges you want to add to the 3D sketch. When you have specified all the included geometry, press [Esc], or right-click and pick the **Done** option.



Complete the Exercise on the Student CD.

Sketching 3D Intersections

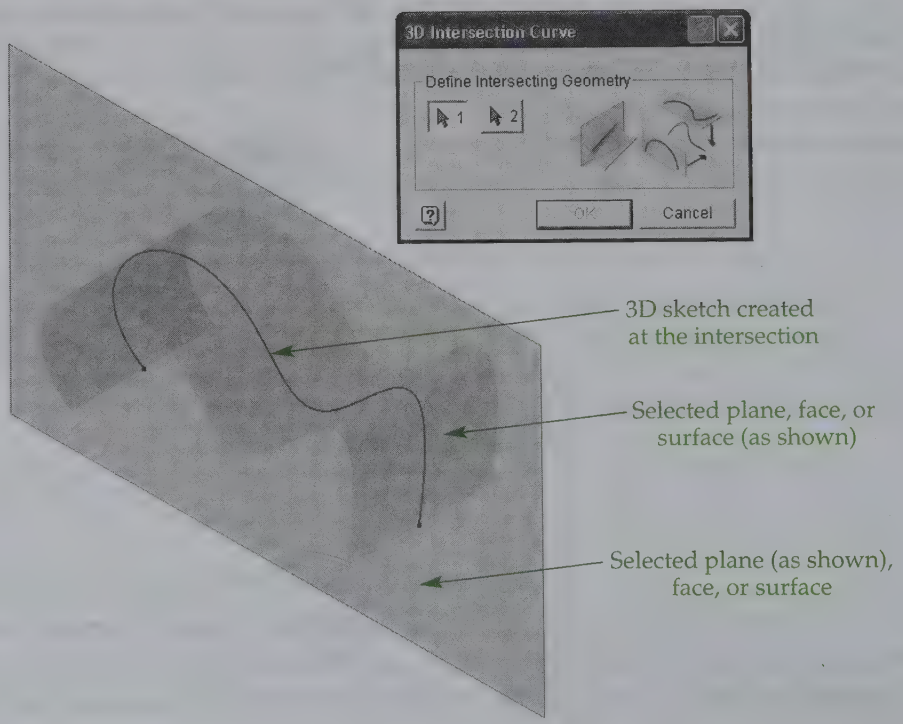
Another way to generate a 3D sketch is to use the **3D Intersection** tool to place a sketch at the intersection of surfaces, planes, and faces. See Figure 9-28. Once you create two intersecting surfaces, planes, or faces, or any intersecting combination of these features, a sketch can be created along the intersection with the **3D Intersection** tool by picking the **3D Intersection** button on the **3D Sketch** panel bar or **3D Sketch** toolbar.

The **3D Intersection Curve** dialog box is displayed when you access the **3D Intersection** tool. To create a 3D sketch at an intersection, as shown in Figure 9-28, pick the first surface, plane, or feature face, followed by the second surface, plane, or feature face. To redefine the first selection, pick the **1** button and then choose the new surface. To redefine the second selection, pick the **2** button, and then choose the new surface. Once you specify the intersection, pick the **OK** button to generate the sketch.



Complete the Exercise on the Student CD.

Figure 9-28.
Creating a 3D sketch using existing surfaces, planes, and faces and the **3D Intersection Curve** dialog box.



Projecting a Curve to a Surface

The **Project Curve to Surface** tool is used to project a curve, typically a 2D sketch, onto a surface, creating a 3D sketch profile. Once you produce a curve and a surface, a sketch can be projected from the curve to the surface with the **Project Curve to Surface** tool by picking the **Project Curve to Surface** button on the **3D Sketch** panel bar or **3D Sketch** toolbar.

The **Project Curve to Surface** dialog box is displayed when you access the **Project Curve to Surface** tool. See Figure 9-29. The first step to projecting a curve onto a surface is to pick the **Faces** button, if it is not already selected, followed by choosing the surface faces on which the curve will be projected. Next, pick the **Curves** button, and select the curve or curves you want to project onto the selected surface. A curve is usually produced as a 2D sketch. Multiple curve profiles can be selected as long as they are on the same sketch plane. Once you pick the faces and the curves, there are three options for projecting the curve onto the surface.

The first projection option involves projecting the curve onto the surface, using a vector direction. To use this technique, pick the **Project Along Vector** output button, which should be activated by default. Then, with faces and curves already picked, choose the **Direction** button, and select the vector direction. The vector direction can be any existing linear feature edge or a work axis. Pick the **Apply** button to project the curve and remain in the **Project Curve to Surface** tool, or select the **OK** button to project the curve and exit the tool. See Figure 9-30.

The second option projects the curve onto the surface according to the closest distance from the curve to the surface. To use this method pick the **Project to Closest Point** output button. Then, with faces and curves already selected, pick the **Apply** button to project the curve and remain in the **Project Curve to Surface** tool, or select the **OK** button to project the curve and exit the tool. See Figure 9-31.

The third option wraps the curve along and around the surface based on the tangent relationship of the surface and the curve. To use this method pick the **Wrap to Surface** output button. Then, with faces and curves already selected, pick the **Apply** button to project the curve and remain in the **Project Curve to Surface** tool, or select the **OK** button to project the curve and exit the tool. See Figure 9-32.

Figure 9-29.
The **Project Curve to Surface** dialog box.

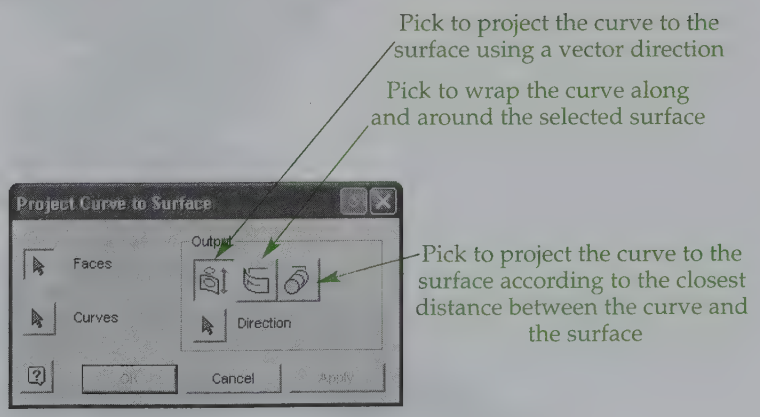


Figure 9-30.
Projecting a curve onto a surface using the **Project Along Vector** output method.

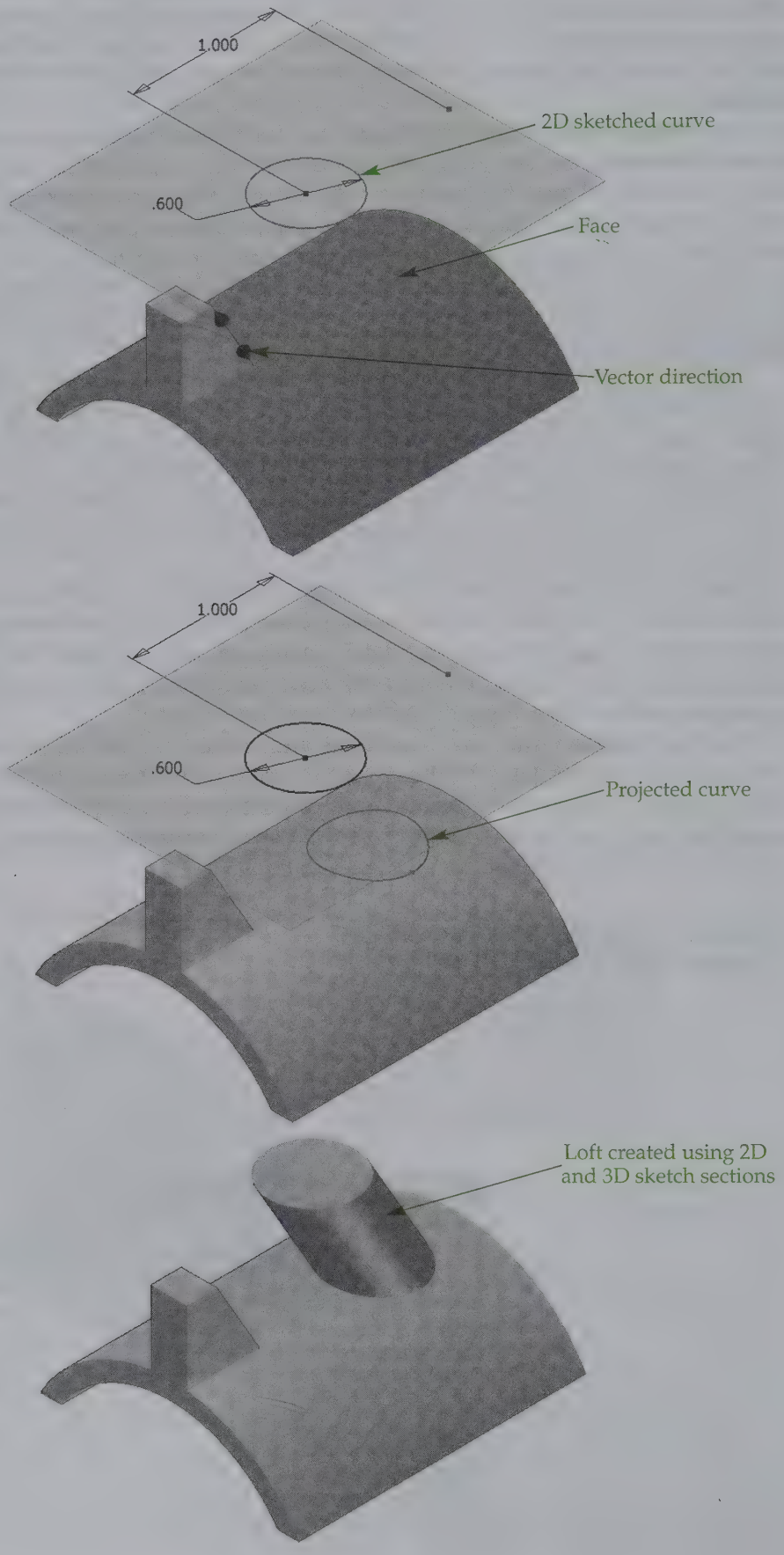


Figure 9-31.
Projecting a curve onto a surface using the **Project to Closest Point** output method.

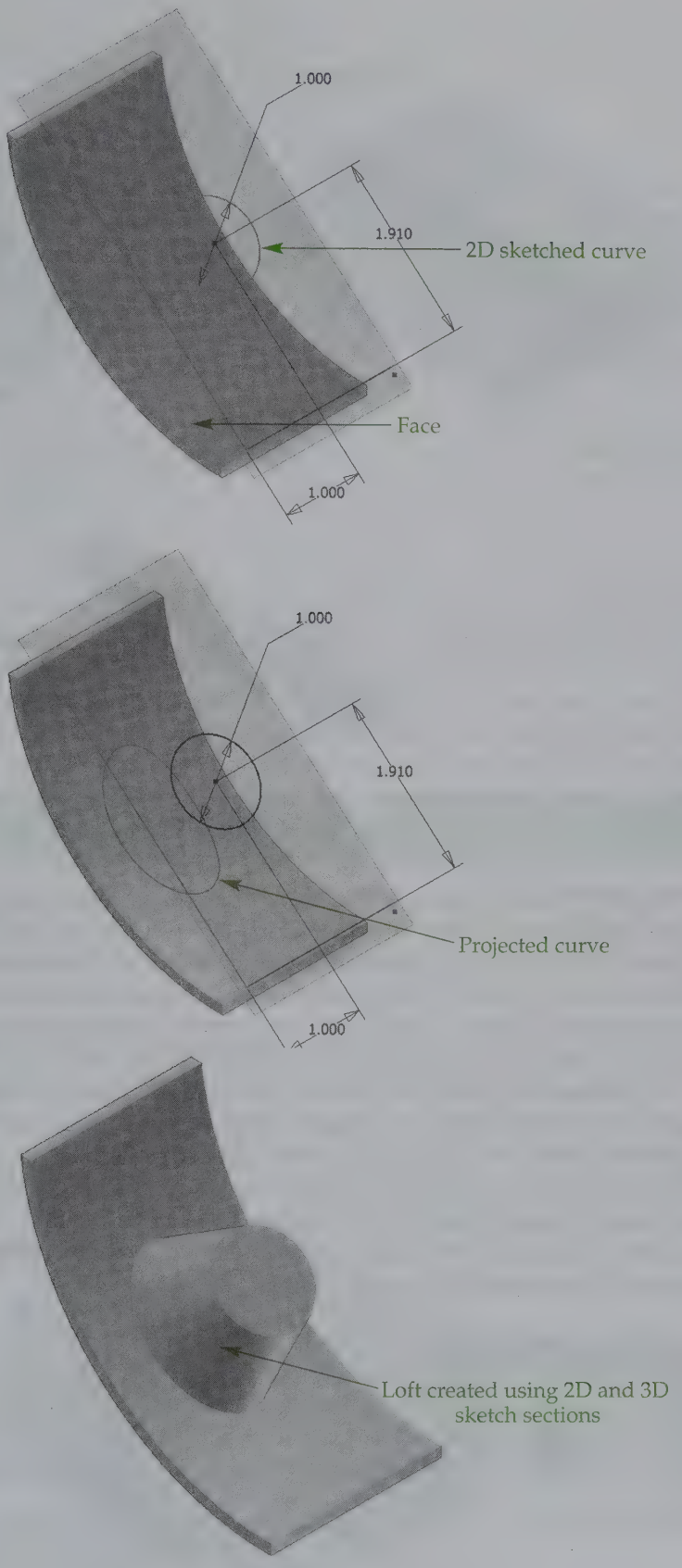
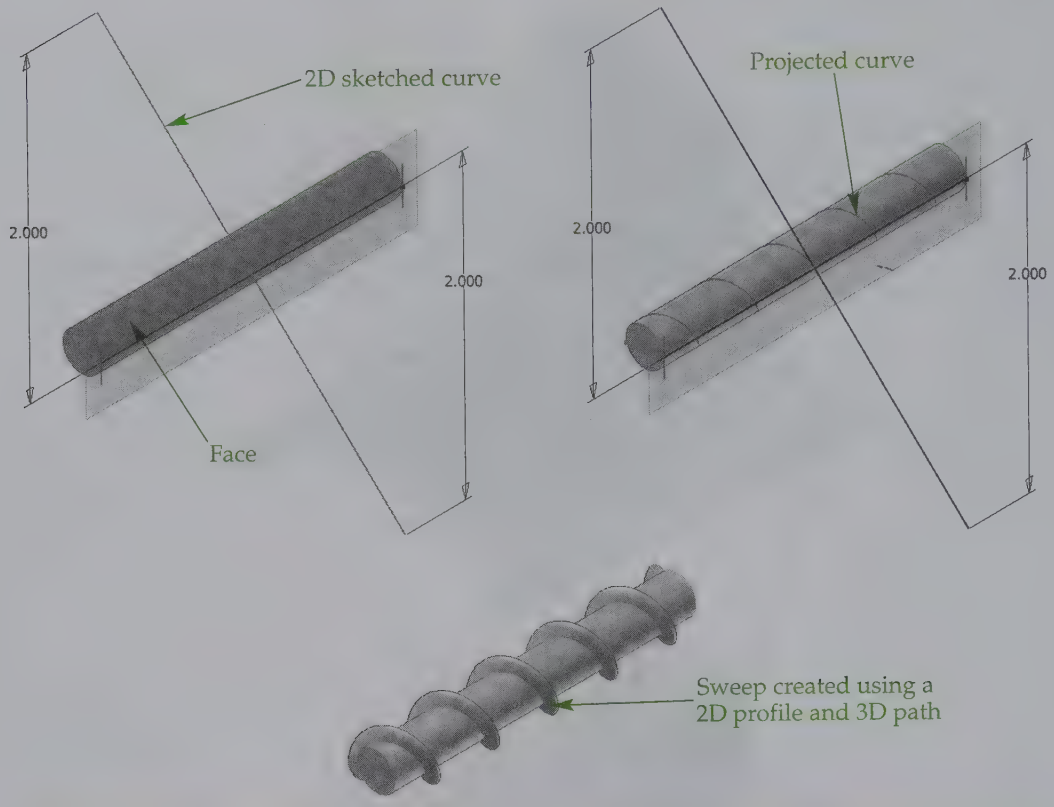


Figure 9-32.
Projecting a curve onto a surface using the **Wrap to Surface** output method.



Creating Split Features

A ***split*** divides a feature into two separate pieces, referencing an existing feature and an additional separation sketch or plane. Splits are often used for casting, forging, and molding applications, but you can find a number of uses for the **Split** tool. Inventor allows you to split a feature two different ways, depending on the purpose. One option is to keep existing feature geometry while splitting feature faces. Another option is to remove a section of an existing feature. See Figure 9-33.

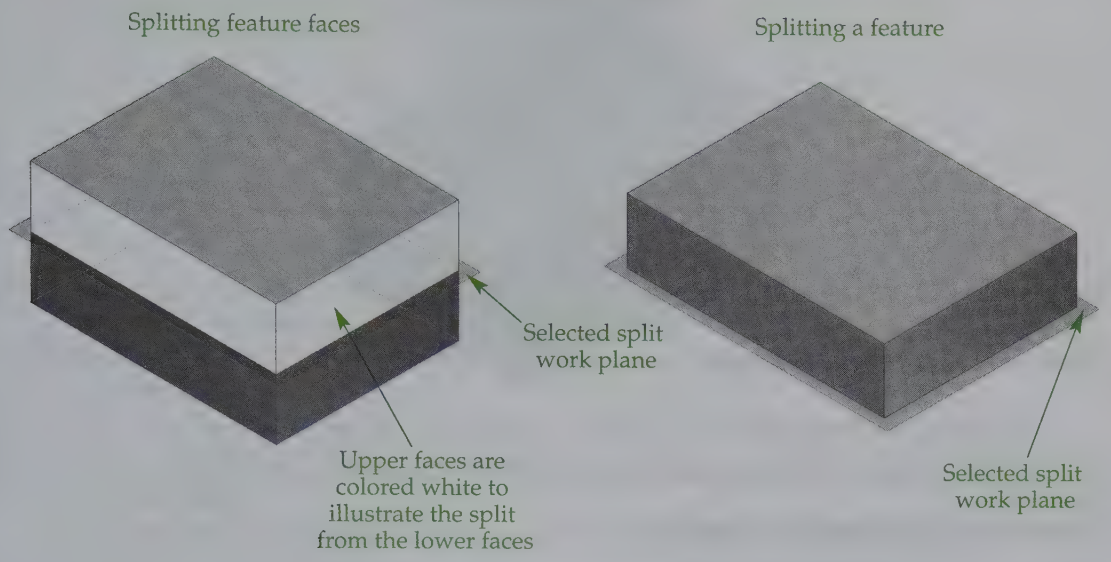
Regardless of the technique you use to split a feature, you must create an additional sketch, plane, face, or surface, which acts as the separation area. To create a parting line or plane for a split add a work plane to the feature, as shown in Figure 9-33; open a 2D sketch on a face or work plane, and sketch geometry using any of the 2D tools, as shown in Figure 9-34; or open a 2D sketch on a face or work plane, and sketch geometry using any of the 2D tools, and then extrude the sketch using the **Surface** operation type option to create a plane. See Figure 9-35.

PROFESSIONAL TIP



When using the **Spline** tool to create a splitting line, you may want to place and constrain points using the **Point, Hole Center** tool. Then access the **Spline** tool, and connect the points.

Figure 9-33.
When splitting a feature, the faces can be split or the entire feature can be split with a portion removed.



Split features are created using the **Split** tool. To access the **Split** tool, pick the **Split** button on the **Part Features** panel bar, or pick the **Split** button on the **Part Features** toolbar. When you access the **Split** tool, the **Split** dialog box is displayed. See **Figure 9-36.** By default, the **Split Face** button is active, which allows you to split feature faces without removing a section of the feature. To use the **Split Face** option, pick the **Split Tool** button if it is not already activated, and select the parting line or plane. The next step is to specify the faces you want to split. By default, the **Select** button is active, which allows you to select the specific faces you want to split. The **Split Face** tool is useful for certain applications, such as when you want to apply different draft angles to various split faces. If you only want to split certain faces, the **Select** button and **Faces to Split** buttons must be active. Pick the faces you want to split, followed by the **OK** button. If you want to split all the faces around the entire feature, select the **All** button, followed by the **OK** button.

Figure 9-34.
Creating a split parting line using a 2D sketch and 2D sketch tools. A line was used in this example.

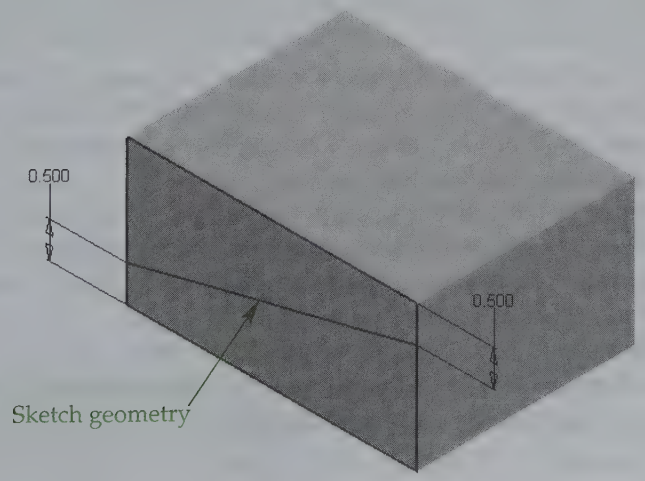


Figure 9-35.
Creating a split parting line by surface-extruding 2D sketch geometry. A spline was used in this example.

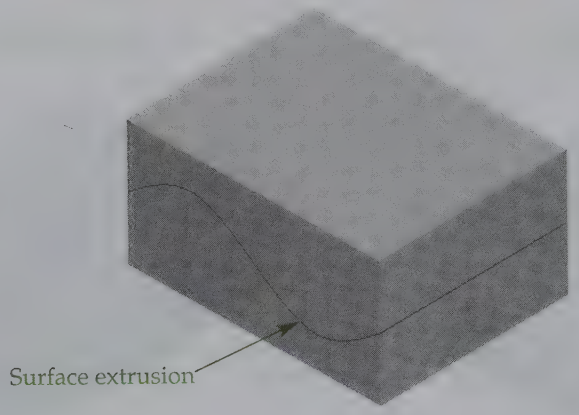
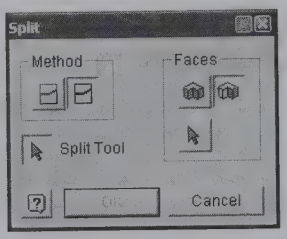


Figure 9-36.
The **Split** dialog box, **Split Face** method.



NOTE

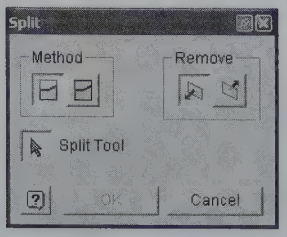
You can only directly access split faces using the **Face Draft** tool. In order to use the split faces for other purposes, such as developing sketch geometry, you must open a new sketch on the face, and project the split edges.

To remove a section of a feature using the **Split** tool, pick the **Split Part** button of the **Split** dialog box. See Figure 9-37. To use the **Split Part** option, pick the **Split Tool** button if it is not already activated. Then, select the parting line or plane. The next step is to specify which section of the part you want to remove by picking one of the **Side to Remove** buttons. Finally, pick the **OK** button to remove the feature section.

Typically, you may want to develop two separate parts when using the **Split Part** option. For example, you may want to create a top half and a bottom half. The two halves should be separate part files, such as Top.ipt and Bottom.ipt. You can then place both parts into an assembly file to make an assembly. To create two part files, use the following steps before you use the **Split** tool:

1. Save the current part with both sections of the part in one piece and the parting line created. The name of the file should be Top.ipt, for example.
2. Using the **Split** tool, remove a section of the part. Using the previous example, you should remove the bottom section.
3. Select **Save Copy As...** from the **File** pull-down menu and save a copy of the current part with one of the sections removed by the **Split** tool. Using the previous example, you should save the copy as Bottom.ipt.

Figure 9-37.
The **Split** dialog box, **Split Part** method.



4. The initial file, in this example, Top.ipt, should currently be opened. Save and close the active file when you are finished creating the part.
5. Open the saved copy of the original part. Using the previous example, you should open Bottom.ipt.
6. Right-click the split in the browser, and select the **Edit Feature** menu option.
7. Select the opposite **Side to Remove** button to flip the section to be removed, and pick the **OK** button.
8. Save and close the active file when you are finished creating the part.

NOTE

The steps previously described for developing two separate "halves" of a part represent one possible technique. You may develop your own techniques depending on the application.

Exercise 9-7 Complete the Exercise on the Student CD.

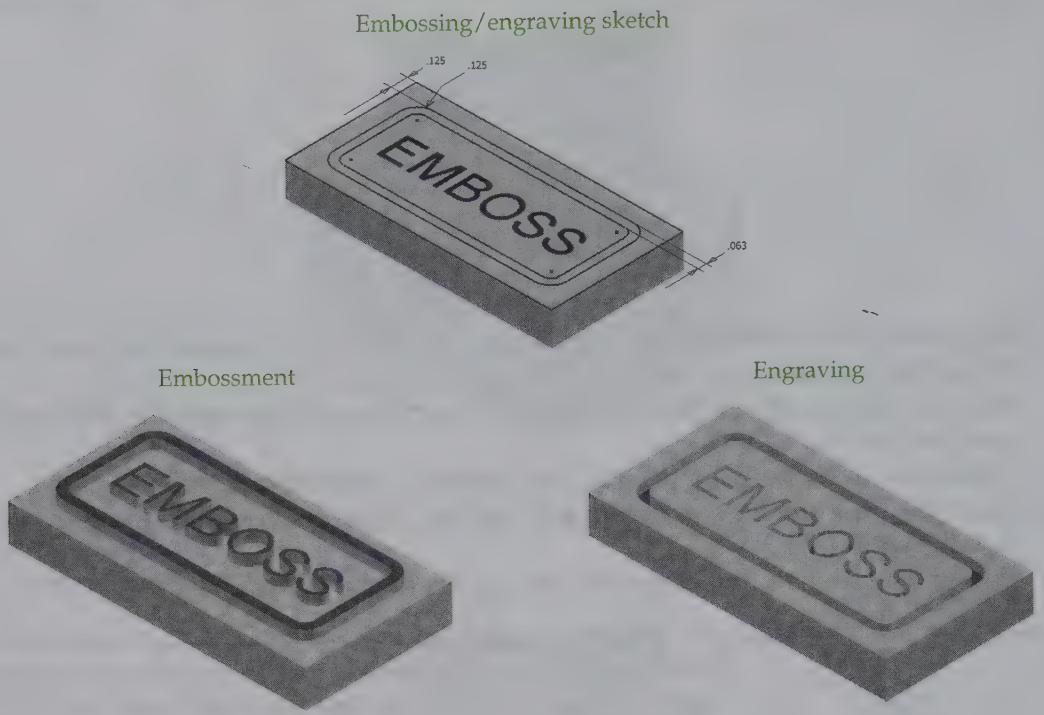
Exercise 9-8 Complete the Exercise on the Student CD.

Embossing and Engraving

Embossing is the process of raising shapes or text off the surface of an object that has volume, such as a block. ***Engraving*** is the opposite of embossing, in that engraving cuts into, or impresses shapes or text into the surface of an object that has volume. Inventor allows you to emboss and engrave sketched geometry or text using the **Emboss** tool, which functions similar to **Extrude**.

The **Emboss** tool can be used to emboss or engrave any sketch geometry onto an existing feature, including text. Figure 9-38 shows a sketch that is used to create an embossing and an engraving. An embossing or engraving sketch can be placed on an existing feature face or on a work plane offset from a feature. When you sketch on an offset work plane, the embossing or engraving is projected onto the feature, even though the sketch may not actually be on the feature face. See Figure 9-39. Once you develop an embossing or engraving sketch, access the **Emboss** tool by picking the

Figure 9-38.
A sketch can be used to create an embossment or an engraving.



Emboss button on the **Part Features** panel bar, or pick the **Emboss** button on the **Part Features** toolbar. The **Emboss** dialog box is displayed when you access the **Emboss** tool. See Figure 9-40.

To emboss or engrave a sketch, first pick the **Profile** button if it is not already selected. Next, pick the sketch profile or profiles you want to emboss from, or engrave into, the feature. Once the sketch profile has been identified, use the following options in the **Emboss** dialog box to complete the feature:

Figure 9-39.
An embossing sketch can be created on an offset work plane to emboss a nonplanar face.

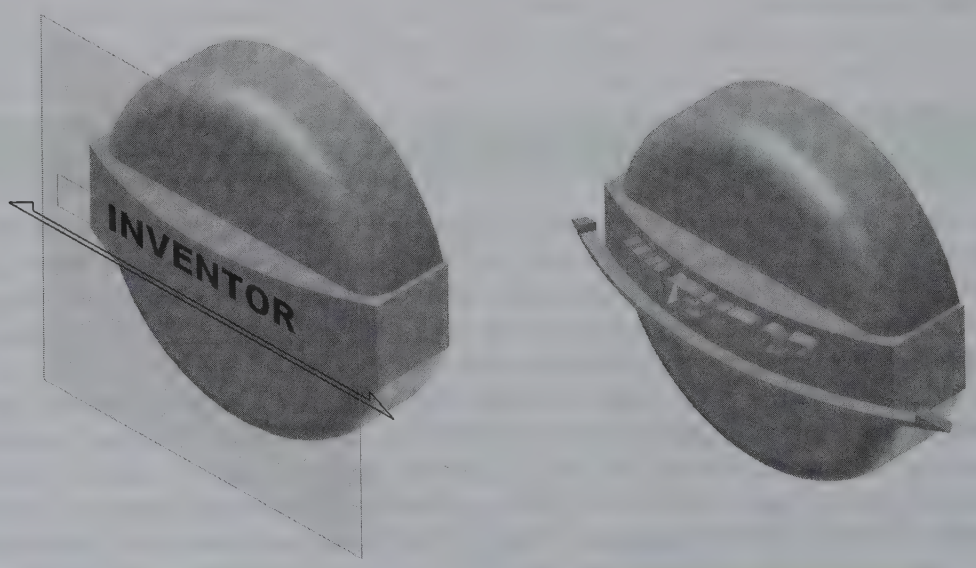
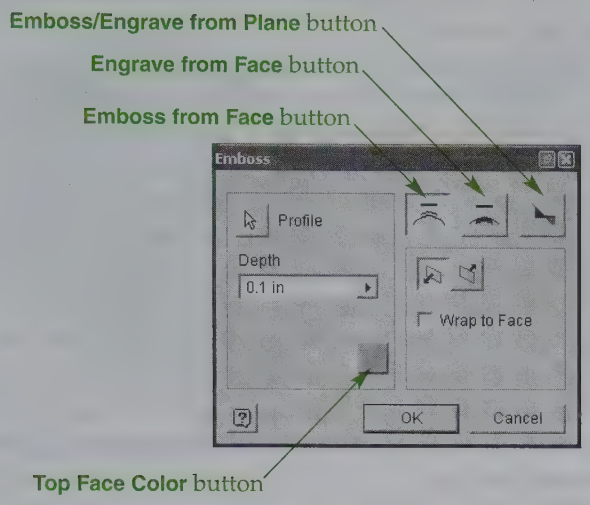


Figure 9-40.
The **Emboss** dialog box, **Emboss from Face** option.



- **Emboss from Face**. Pick this button to create an embossed feature. Again, an embossing raises the selected sketch geometry away from a feature, much like a join extrusion. Once you select the **Emboss from Face** button, enter or select the embossing depth using the **Depth** edit box, and if needed, define the embossing direction by picking the required **Flip** button. By default, if you create an embossing from a curved face, the sketch profile is embossed straight out of the face. However, if you want the embossing to wrap around the curved face pick the **Wrap to Face** check box, and select the curved face. See **Figure 9-41**. The last option for embossing from a face is to add color to the top face of the embossing

Figure 9-41.
Compare embossing without wrapping to a curved face and wrapping to a curved face.

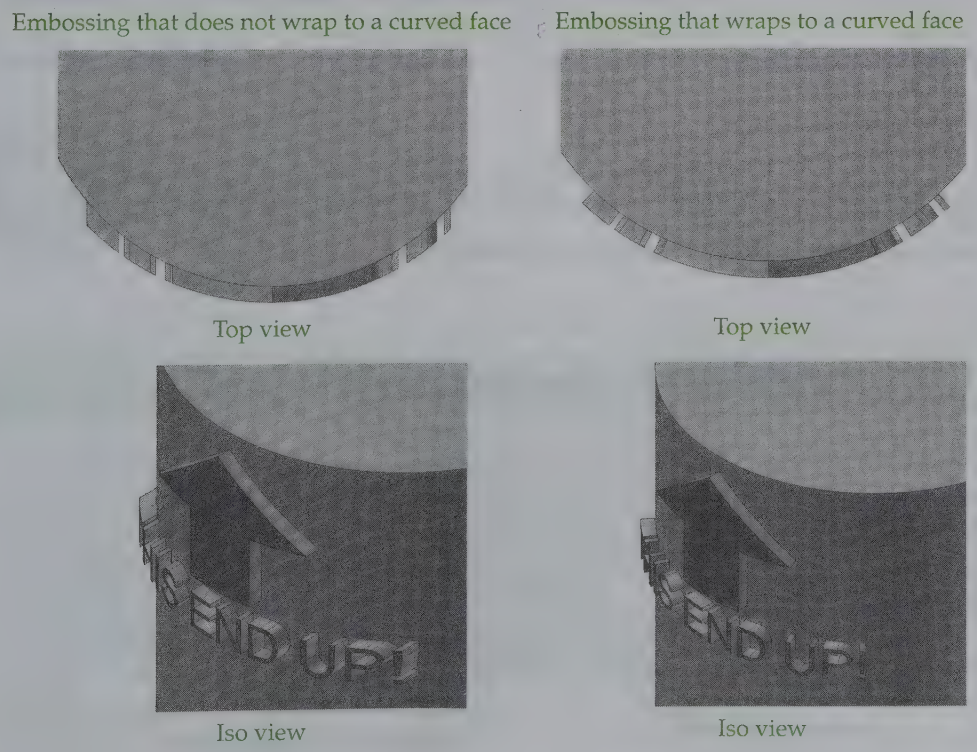
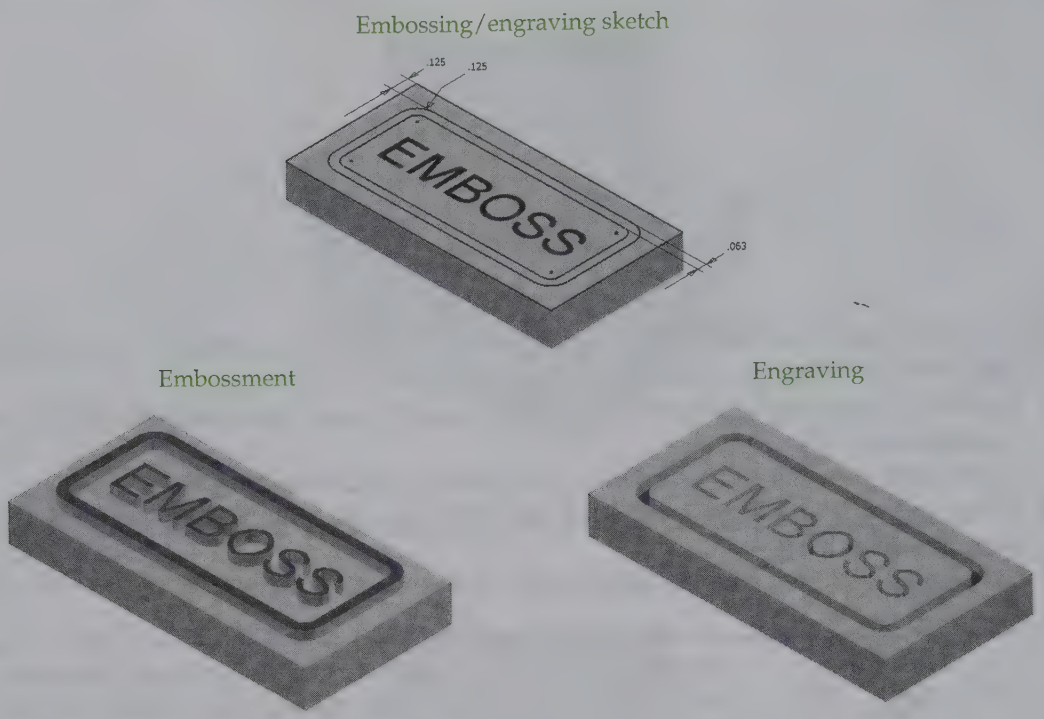


Figure 9-38.
A sketch can be used to create an embossment or an engraving.



Emboss button on the **Part Features** panel bar, or pick the **Emboss** button on the **Part Features** toolbar. The **Emboss** dialog box is displayed when you access the **Emboss** tool. See Figure 9-40.

To emboss or engrave a sketch, first pick the **Profile** button if it is not already selected. Next, pick the sketch profile or profiles you want to emboss from, or engrave into, the feature. Once the sketch profile has been identified, use the following options in the **Emboss** dialog box to complete the feature:

Figure 9-39.
An embossing sketch can be created on an offset work plane to emboss a nonplanar face.

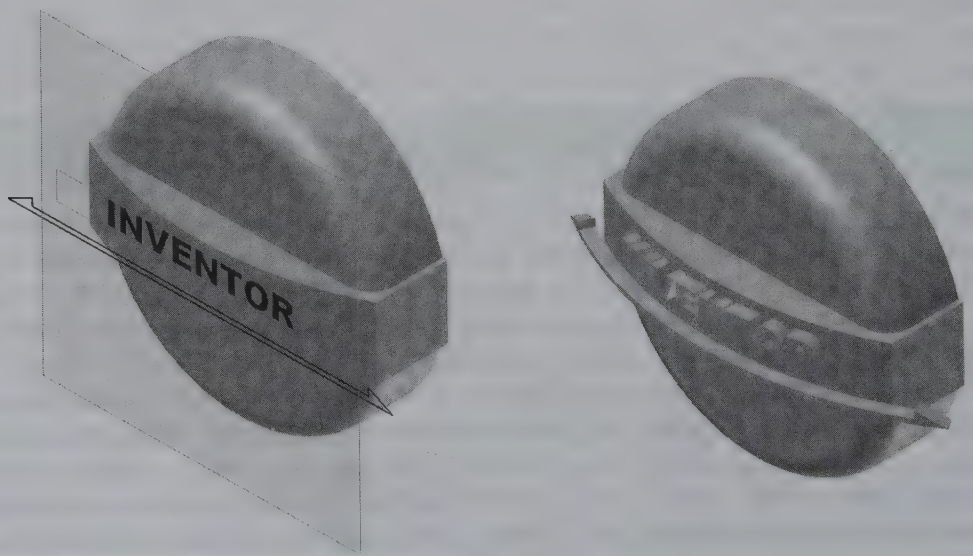
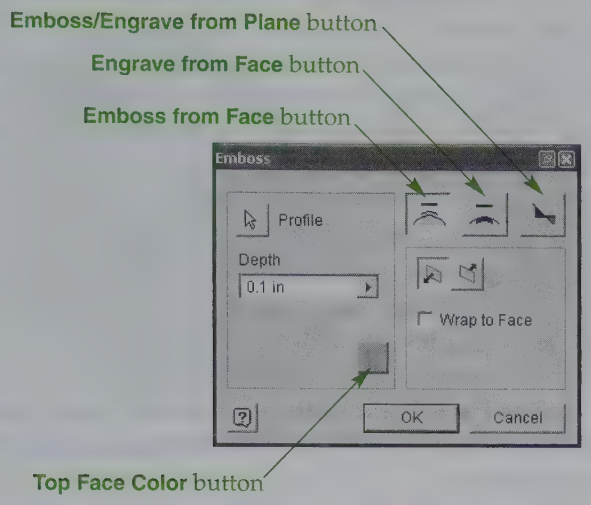
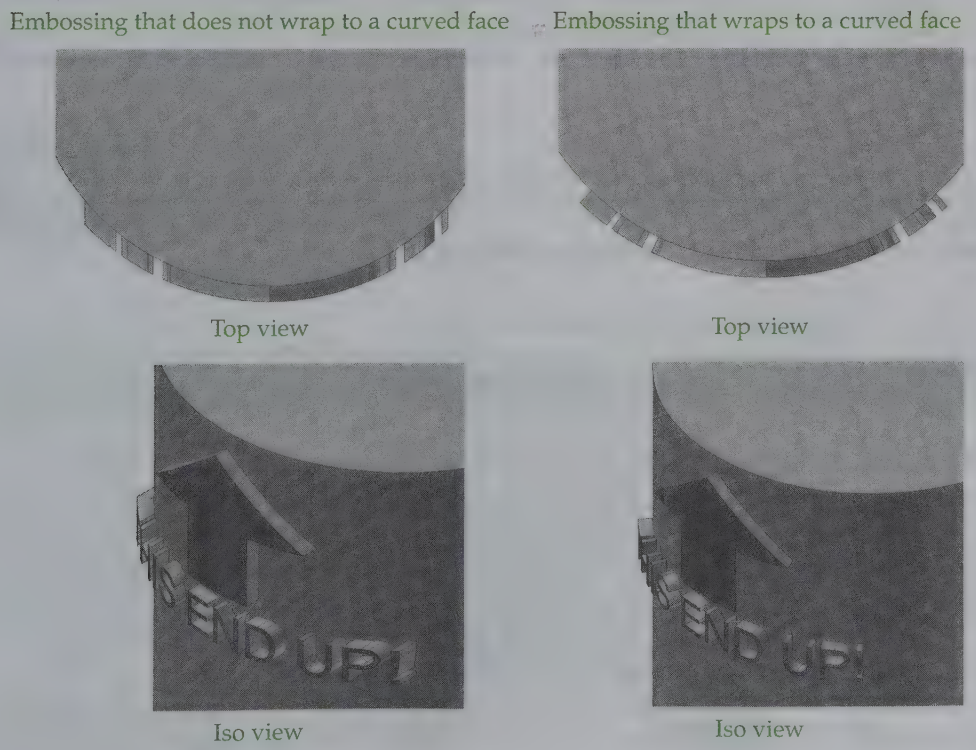


Figure 9-40.
The **Emboss** dialog box, **Emboss from Face** option.



- **Emboss from Face**. Pick this button to create an embossed feature. Again, an embossing raises the selected sketch geometry away from a feature, much like a join extrusion. Once you select the **Emboss from Face** button, enter or select the embossing depth using the **Depth** edit box, and if needed, define the embossing direction by picking the required **Flip** button. By default, if you create an embossing from a curved face, the sketch profile is embossed straight out of the face. However, if you want the embossing to wrap around the curved face pick the **Wrap to Face** check box, and select the curved face. See **Figure 9-41**. The last option for embossing from a face is to add color to the top face of the embossing

Figure 9-41.
Compare embossing without wrapping to a curved face and wrapping to a curved face.



by picking the **Top Face Color** button to access the **Color** dialog box. Select the desired color from the **Color** drop-down list and pick the **OK** button. Once you fully define the embossing using the options in the **Emboss** dialog box, pick the **OK** button to generate the feature.

- **Engrave from Face**. Pick this button to create an engraved feature. Again, an engraving cuts into a feature using the selected sketch geometry, much like a cut extrusion. Once you select the **Engrave from Face** button, enter or select the engraving depth using the **Depth** edit box, and if needed, define the engraving direction by picking the required **Flip** button. By default, if you create an engraving into a curved face, the sketch profile is engraved straight into the face, much like the embossing that does not wrap in Figure 9-41. If you want the engraving to be wrapped around the curved face, similar to the embossing that does wrap in Figure 9-41, pick the **Wrap to Face** check box and select the curved face. Just as when embossing from a face, you can add color to the engraved face using the **Top Face Color** button and **Color** dialog box. Once you fully define the engraving using the options in the **Emboss** dialog box, pick the **OK** button to generate the feature.
- **Emboss/Engrave from Plane**. Pick this button if you want to emboss or engrave the selected sketch profile from the sketch plane at a specified taper angle. When you pick the **Emboss/Engrave from Plane** button, the **Emboss** dialog box displays the options shown in Figure 9-42. To apply an embossing/engraving from the sketch plane, enter or select a draft taper angle from the **Taper** edit box. Then, if needed, define the engraving direction by picking the required **Flip** button. Figure 9-43 shows examples of using different flip options and taper angles. Just as when embossing from a face or engraving from a face, you can add color to the embossed/engraved face using the **Top Face Color** button and **Color** dialog box. Once you fully define the embossing/engraving using the options in the **Emboss** dialog box, pick the **OK** button to generate the feature.

NOTE

If the sketch you create extends past a curved face, you may have to pick the **Wrap to Face** check box in order to create an embossing or engraving.

Figure 9-42.
The **Emboss** dialog box, **Emboss/Engrave from Plane** option.

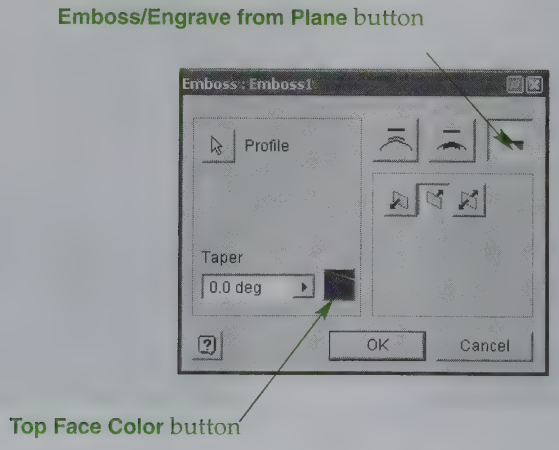
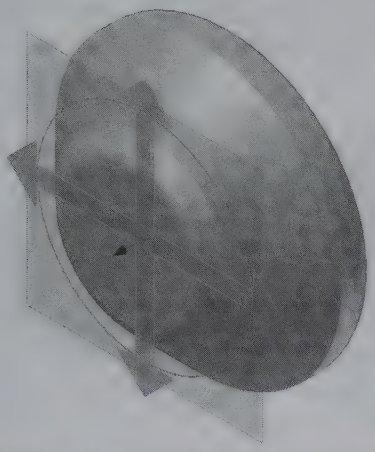
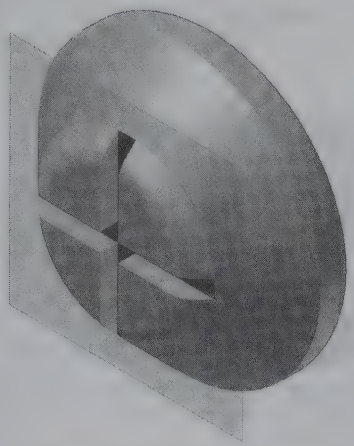


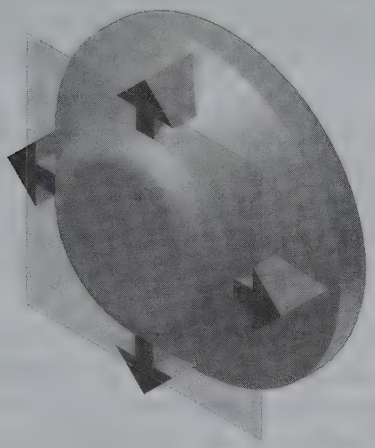
Figure 9-43.
These parts started with the same sketch profile, but with different emboss/engrave settings.



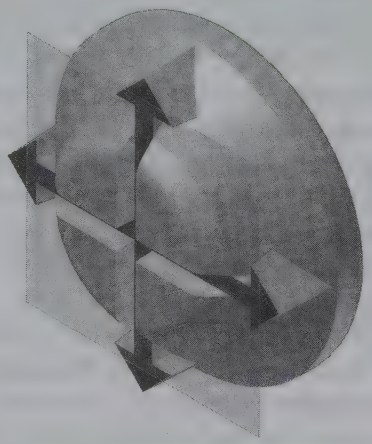
Emboss/engrave from plane, sketch



Embossing/engraving toward the model, with a taper of 0°



Embossing/engraving away from the model, with a taper of 15°



Embossing/engraving in both directions, with a taper of 10°



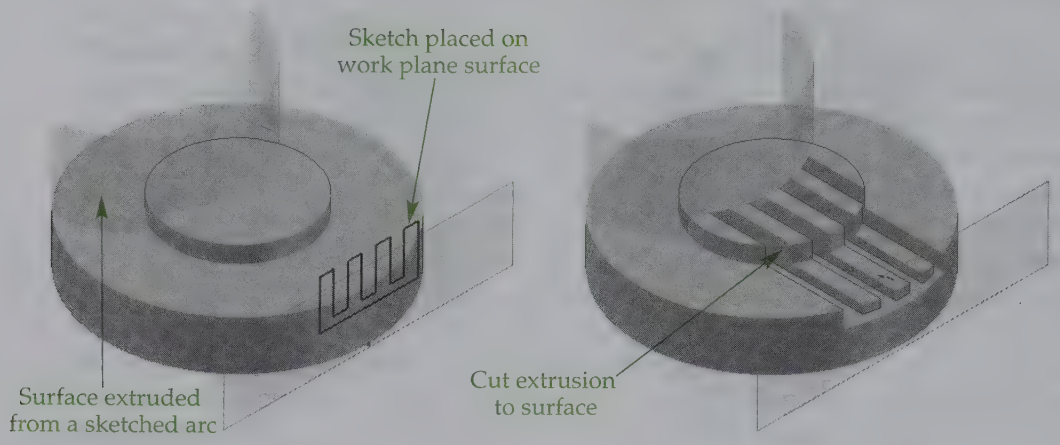
Exercise 9-9 Complete the Exercise on the Student CD.

Additional Surface Tools and Techniques

A ***surface*** is a volumeless shape that is primarily used for construction purposes, allowing you to generate more advanced model geometry. You have already discovered how to create surfaces using a variety of tools including the **Extrude**, **Revolve**, **Loft**, **Sweep**, and **Thicken/Offset** tools. However, unless you have used other CADD programs in which surfaces are common items, you may still not fully understand how to work with surface features to create complex model shapes.

Like work planes, surfaces help you develop model geometry that is difficult, or impossible, to produce otherwise. In addition, surface visibility can be turned on and off as required, and surfaces take up less space on your computer than solids. Figure 9-44 shows

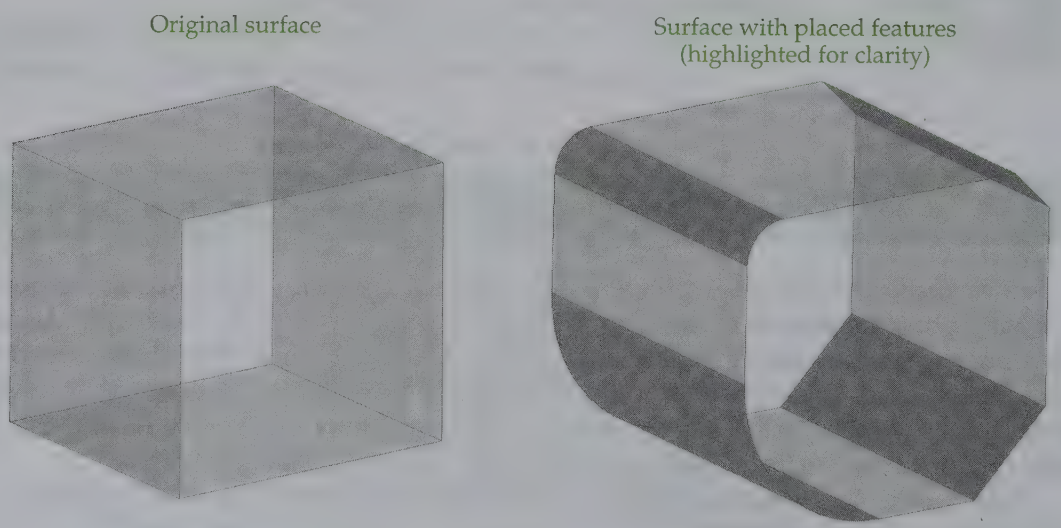
Figure 9-44.
An example of using a surface to control the shape of an extrusion.



one of the most common reasons for creating a surface—to specify a more complicated termination area for other features. You can find a potentially unlimited number of uses for surfaces depending on the purpose. Regardless of the application, use the following information when developing surface features:

- Surfaces can be created using closed loop sketches, such as a rectangle or circle, or open loop sketches, such as a line or arc. Surface sketches can also be modified just as any other feature sketch.
- Surfaces are displayed and labeled accordingly in the browser.
- You can add placed features like fillets and chamfers to surface edges, such as those shown in **Figure 9-45**.
- If you want surfaces to be opaque, just as with other model geometry, instead of translucent, pick the **Opaque Surfaces** check box in the **Construction** area of the **Options** dialog box, **Part** tab. You can also change the display of individual surfaces by right-clicking the desired surface in the browser and deselecting the **Translucent** pop-up menu option.

Figure 9-45.
Adding placed features, radii and chamfers, to a surface.



- To turn off the visibility of, or hide, a surface, right-click on the surface and deselect the **Visibility** pop-up menu option.
- Surfaces can be patterned using feature pattern tools.

The following additional surface modeling tools and techniques are found in Appendix D on the CD accompanying this textbook:

- Replacing faces.
- Sculpting surfaces.
- Deleting faces, lumps, and voids.
- Creating boundary patches.
- Trimming and extending surfaces.
- Stitching and promoting surfaces.

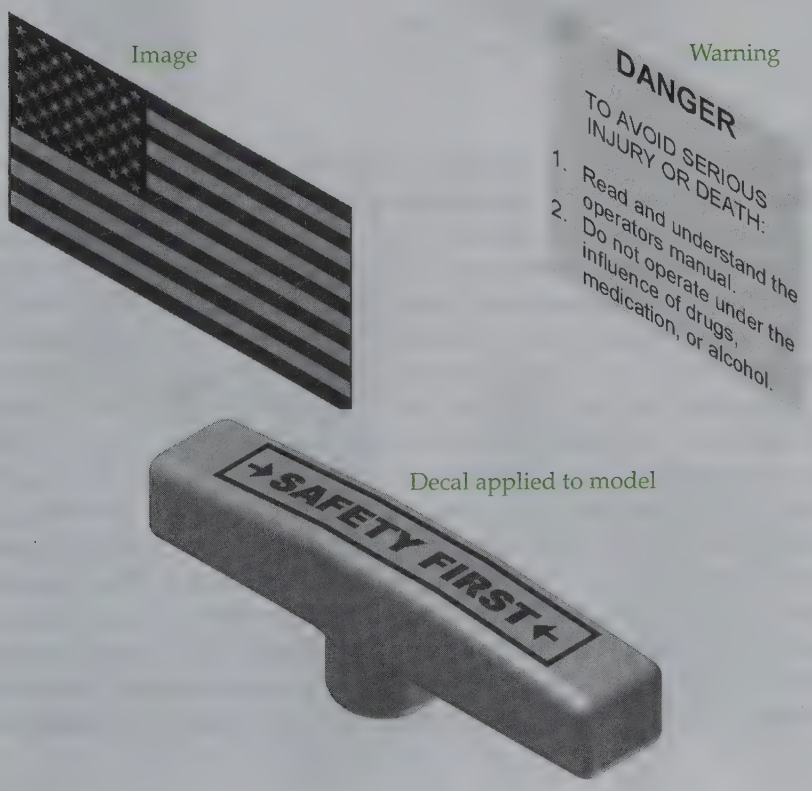


Exercise 9-10 Complete the Exercise on the Student CD.

Creating Decals

A ***decal*** is any type of image applied to a part or assembly and is used to display information or decorate a product. Decals are used in a variety of applications such as a product label, tag, art, text, or seal. See **Figure 9-46.** Decals are sketched features created by inserting an image, text, or spreadsheet file into a part sketch using the **Insert Image** tool. Then the part feature **Decal** tool is used to define the sketched image as a decal

Figure 9-46.
Decal examples.



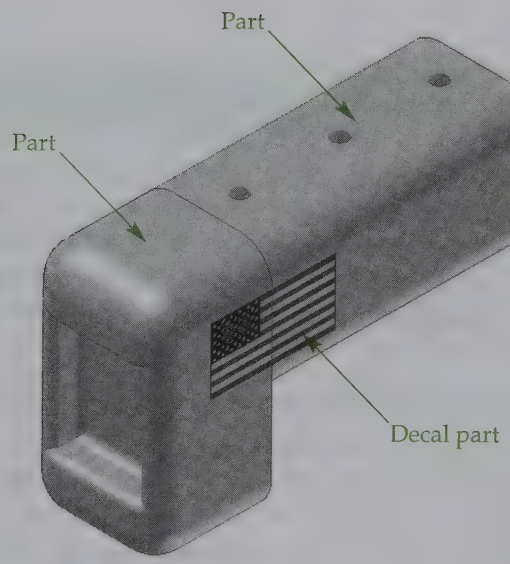
feature on a selected feature face. Though a decal can represent a surface finish, similar to applying paint, in many applications a decal is actually a separate part with its own properties, including a part number, and should be created in a separate part file. Once a decal is created as a single part, it can be placed into multiple assemblies and constrained to other assembly components. For example, **Figure 9-47** shows an assembly of three individual parts. One of the components is a decal that has been placed on the other two parts.

NOTE



Assemblies are described later in this text. However, you can create a decal without working with assemblies.

Figure 9-47.
Using a decal as a separate part.



To add a decal to a part, you must first create the part features on which the decal is placed. When applying a decal to an existing part, the features are already available. See **Figure 9-48**. However, if you plan to create a single part decal that can be used in assemblies, you must produce some type of thin feature to accept the decal image. The feature can be a thin piece of material that represents the actual thickness of the decal, such as the extruded rectangle, or a surface. See **Figure 9-49**. The feature you create defines the face on which the decal is placed and should be the same size as the decal. If the face is too small, the decal will be cut off around the edges, and if the face is too large, the actual size of the sketched decal image may not be correctly represented.

Once you create the part on which you want to add a decal, you must identify a surface to sketch the decal image. A decal image sketch is produced the same as any other 2D sketch and can be placed on an existing feature face, surface, or work plane. In addition, the decal image sketch can be placed away from the feature with the decal image being projected from the sketch plane onto the feature. See **Figure 9-50**. In fact, it is often easier to generate a decal from a sketch that is offset from the feature face.

Decal sketches are .bmp, .doc, or .xls files inserted onto an open sketch plane using the **Insert Image** tool. Access the **Insert Image** tool by picking the **Insert Image** button

Figure 9-48.
An example of a part that will receive a decal.



Figure 9-49.
Extruding a rectangle to create a part decal on a thin solid and extruding a line to create a part decal on a surface.

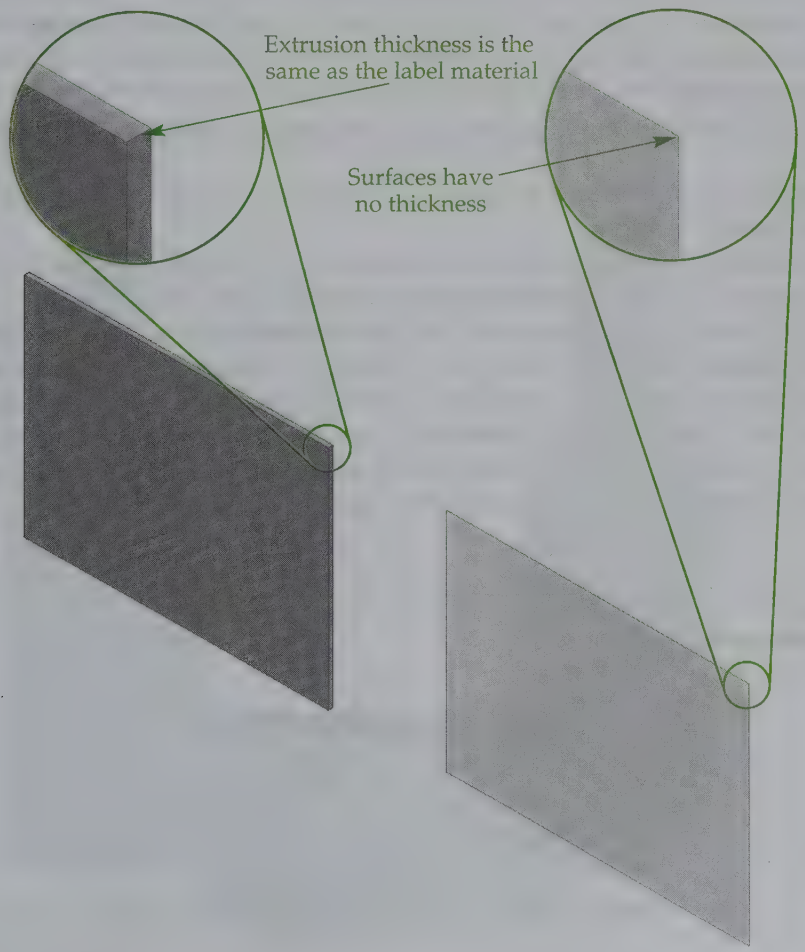
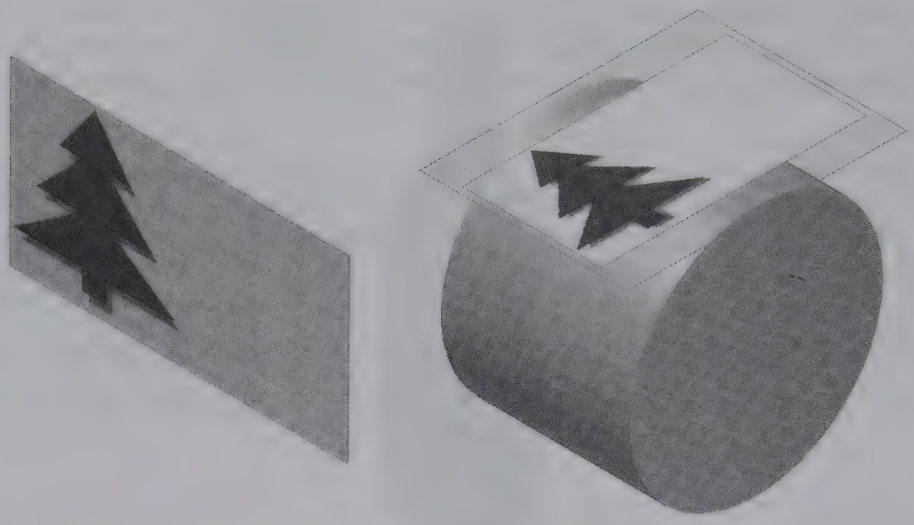


Figure 9-50.
Sketching on an existing face (thin extrusion face) and sketching on a work plane.



Sketching on an existing face

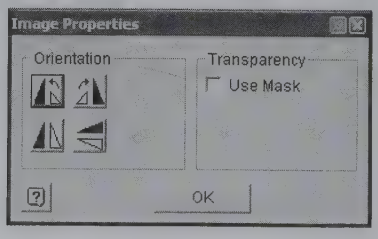
Sketching on a work plane

on the **2D Sketch Panel** bar; picking the **Insert Image** button on the **2D Sketch Panel** toolbar; or selecting **Image...** from the **Insert** pull-down menu.

When you access the **Insert Image** tool, the **Open** dialog box is displayed, allowing you to select the file you want to insert. Once you locate the desired file, pick the **OK** button to insert the image into the graphics window. Pick a location for the image, and continue placing copies of the image if needed. To exit the **Insert Image** tool, right-click and select the **Done** pop-up menu option, or press the [Esc] key on your keyboard. The imported image has a rectangular boundary, which is defined when the image is created. Still, you can use the following techniques to manipulate a variety of image properties:

- Drag an image boundary corner to resize the image, or drag the entire image to a new location.
- Use sketch dimensions and constraints to specify the size and location of the image (though some dimensions are controlled by the image).
- Right-click on the image and pick the **Properties** pop-up menu option to access the **Image Properties** dialog box shown in Figure 9-51. The **Image Properties** dialog box allows you to rotate and mirror the image using the **Rotate** and **Mirror** buttons. These buttons rotate or mirror the image in reference to the specified image dimensions and constraints. In addition, you can pick the **Use Mask** check box to display the image with a transparent background.

Figure 9-51.
The **Image Properties** dialog box.



When you fully define the image and sketch size, orientation, and properties, finish the sketch and enter the part features environment.

Once you create the feature on which you plan to project the decal, and sketch the decal image, you are ready to generate the decal feature using the **Decal** tool. Access the **Decal** tool by picking the **Decal** button on the **Part Features** panel bar, or picking the **Decal** button on the **Part Features** toolbar.

The **Decal** dialog box is displayed when you access the **Decal** tool. See Figure 9-52. Pick the **Select Image** button if it is not already active, and select the decal image sketch. If you are applying the decal to a planar face, with no curves or edges, pick the **OK** button to generate the decal. However, when placing a decal on a curved or irregular shape, the image may not be projected correctly. As a result, you must select the **Wrap to Face** check box in order for the sketch to be folded around nonlinear faces. See Figure 9-53. You can select only one face on which to project the decal image. As a result, if you want the decal to be placed on multiple adjacent faces, such as over an edge or on a chamfer, you must select the **Chain Faces** check box. See Figure 9-54. Finally, pick the **OK** button to generate the decal.

Figure 9-52.
The **Decal** dialog box.

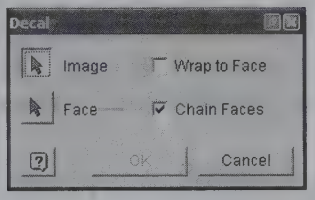


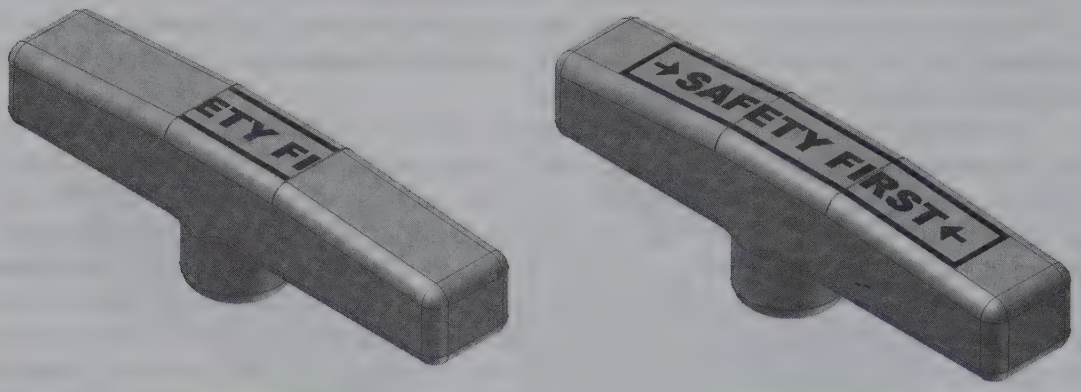
Figure 9-53.
Compare the distortion of a decal placed with and without wrapping to a curved face.



Wrap to Face is off

Wrap to Face is on

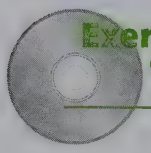
Figure 9-54.
Placing a decal with and without chaining faces.



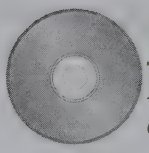
Chain Faces is off

Chain Faces is on

Again, many decals are separate parts and should be inserted into an assembly as an individual component. However, you can add a decal to a completed part, which is similar to painting or changing the color of a part. Typically this technique is used if you do not require an individual part number, do not want to use the decal in different assemblies with multiple components, or want to use the decal only for single part surface display characteristics. The **Decal** tool functions exactly the same whether you generate a separate decal part or place the decal on a finished part.



Exercise 9-11 Complete the Exercise on the Student CD.



Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. Describe a *loft* and how it is similar to an extrusion.
2. What must you do before you can create a loft?
3. Define and explain the function of a *loft section.*
4. What is a *rail*?
5. Describe a *sweep* and tell how it is similar to an extrusion.
6. Briefly explain the difference between a *basic* and *advanced sweep.*
7. Give an example for the use of a *three-dimensional sweep.*
8. What items must exist before you can create a 3D sketch line or spline?
9. Explain why the *line intersection bend* options do not apply to splines?
10. What is a *split* and what does it reference?
11. Give a use for a split.
12. Identify the two different ways Inventor allows you to split a feature.
13. Define *embossing.*
14. Define *engraving.*
15. Briefly define a *decal* and identify a use for one.
16. Identify the file types that can be selected as decal sketches when you access the **Insert Image** tool, allowing you to select the file you want to insert.

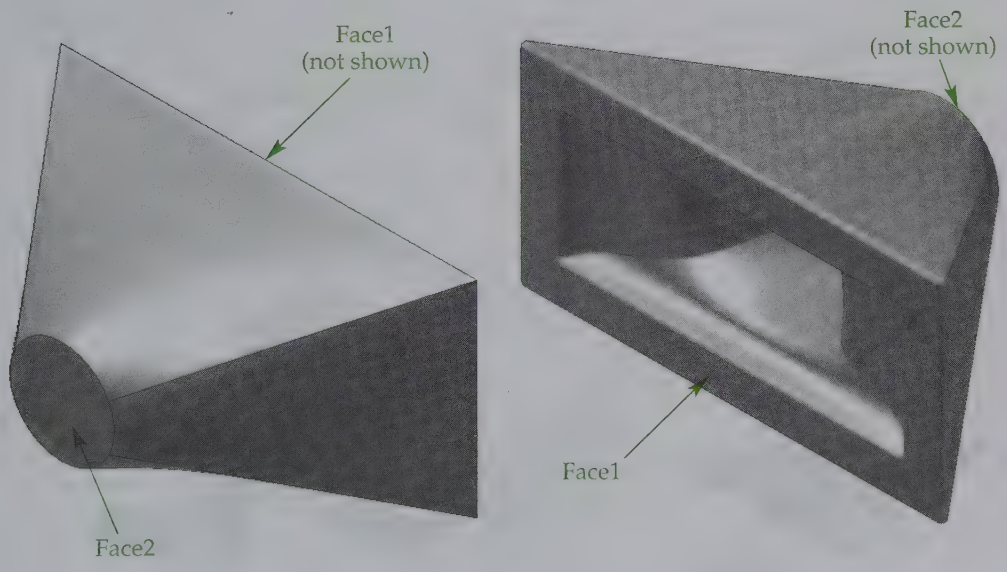
Problems

Instructions:

- Follow the specific instructions provided for each problem.

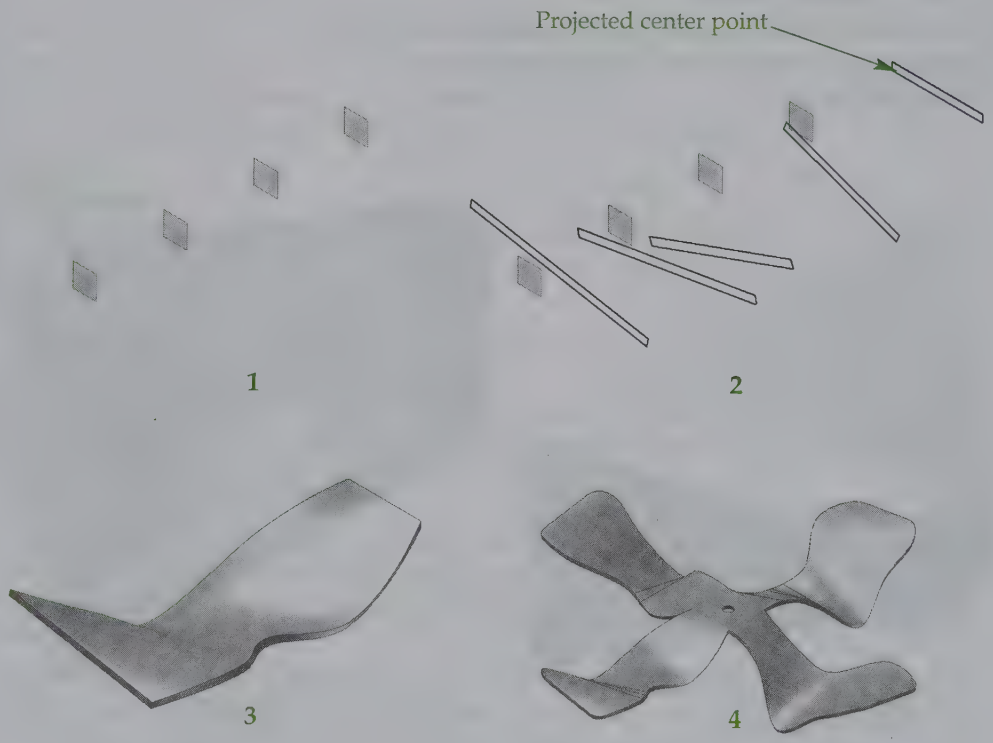
1. **Name:** Nozzle
   Units: Metric
   Template: PART-mm.ipt
   Material: Default
   Color: Default
   Specific Instructions:
   - Open EX9-1.ipt and save a copy as P9-1.ipt.
   - Close EX9-1.ipt without saving and open P9-1.ipt.
   - Open a new sketch on Face1, and offset the projected edges 50 mm.
   - Open a new sketch on Face2, and offset the projected edges 15 mm.
   - Use the **Loft** tool to create a loft feature using the following information:
     - **Sections:** Sketch3 and Sketch4
     - **Operation Type:** Cut
     - **Angle:** 90°
     - **Weight:** 100 ul
   - Add fillets and rounds using a radius of 5 mm.

   The final part should look like the part shown.
   Save As: P9-1.ipt.



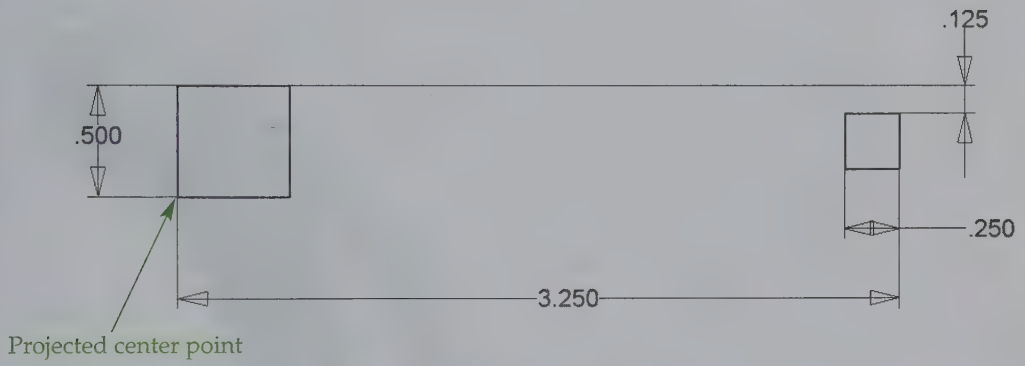
2. **Name:** Tiller Blade
Units: Inch or Metric
Template: PART-in.ipt or PART-mm.ipt
Material: Default or your choice
Color: Default or your choice
Specific Instructions:
- Begin a new inch or metric part file.
- Create a tiller blade similar to the one shown using the following steps:
  A. Create four equally spaced work planes.
  B. Open sketches on the work planes and sketch rectangles at different angles.
  C. Loft the sketches using point mapping.
  D. Fillet the sharp end corners.
  E. Extrude a center support.
  F. Place a hole in the center of the center support.
  G. Circular pattern the loft feature.

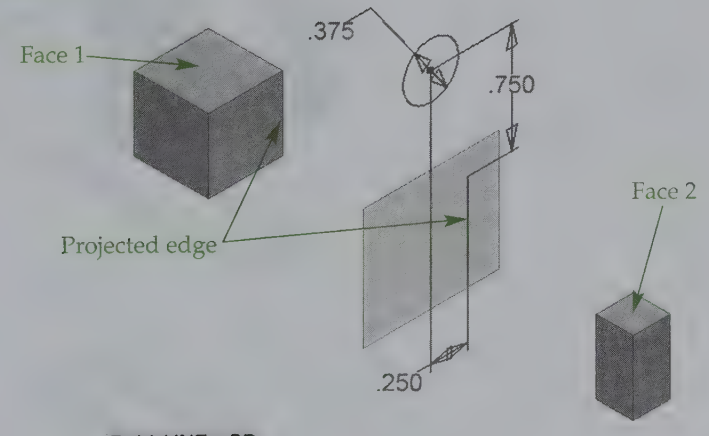
The final part should look similar to the part shown.
Save As: P9-2.IPT.

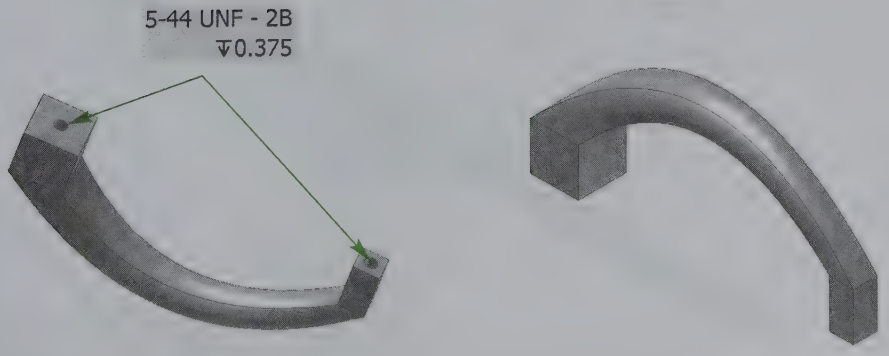


3. **Name:** Handle
Units: Inch
Template: PART-in.ipt
Material: Default
Color: Default
Specific Instructions:
- Begin a new inch part file.
- Create the sketch shown on the XY plane.
- Extrude the sketch .5″ in a positive direction.
- Offset a work plane, 1.75″ from the YZ plane.
- Sketch the geometry shown on the work plane.
- Use the **Loft** tool to create a loft feature shown using the following information:
  - **Sections:** Face 1, followed by Sketch2, followed by Face 2.
  - **Operation Type:** Join
  - **Weight:** 0 ul
- Place the specified holes in the centers of the extrusion bases, as shown.

The final part should look like the part shown.
Save As: P9-3.ipt.

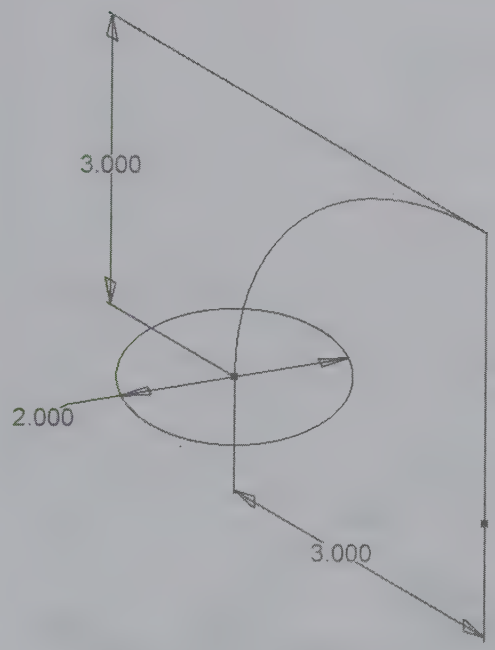






4. **Name:** 90° Reducing Elbow
Units: Inch
Template: PART-in.ipt
Material: Brass, Soft Yellow
Color: Nickel (Bright)
Specific Instructions:
- Begin a new inch part file.
- Sketch a ∅2" circle around the projected center point on the XY plane.
- Open a new sketch on the XZ plane, and sketch the arc shown.
- Use the **Sweep** tool to create a sweep feature using the following information:
  - **Operation Type:** Join
  - **Taper:** –6°
- Shell the sweep feature to create .125" thick walls, and remove the end faces.
- Add .1" chamfers to the end edges.

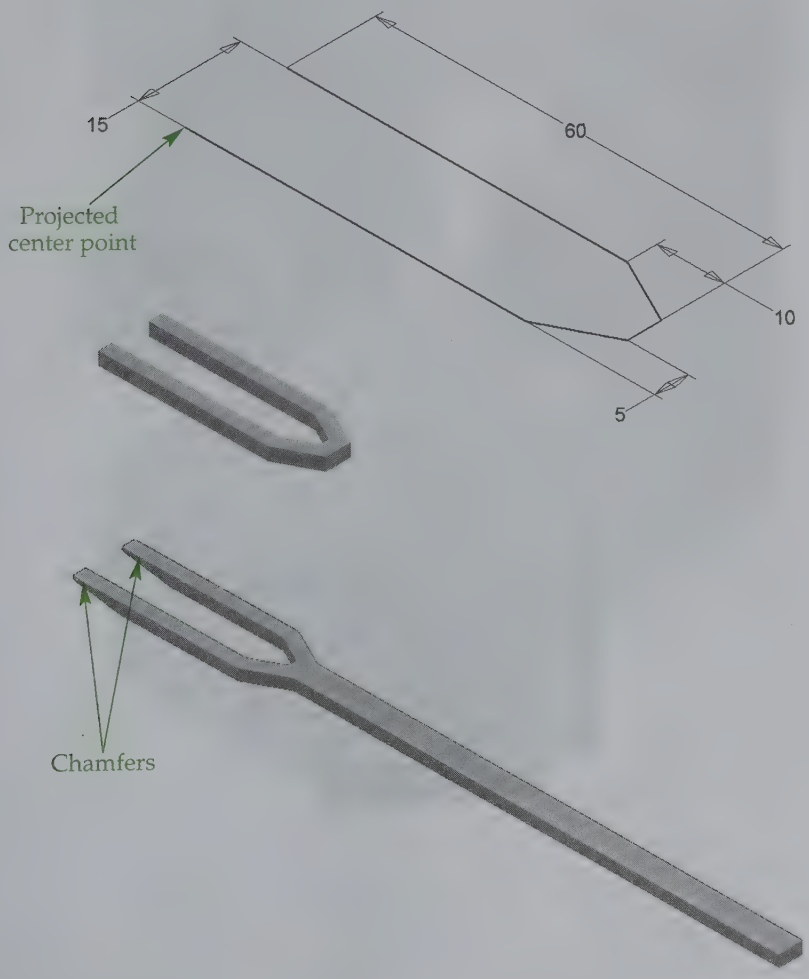
The final part should look like the part shown.
Save As: P9-4.ipt.





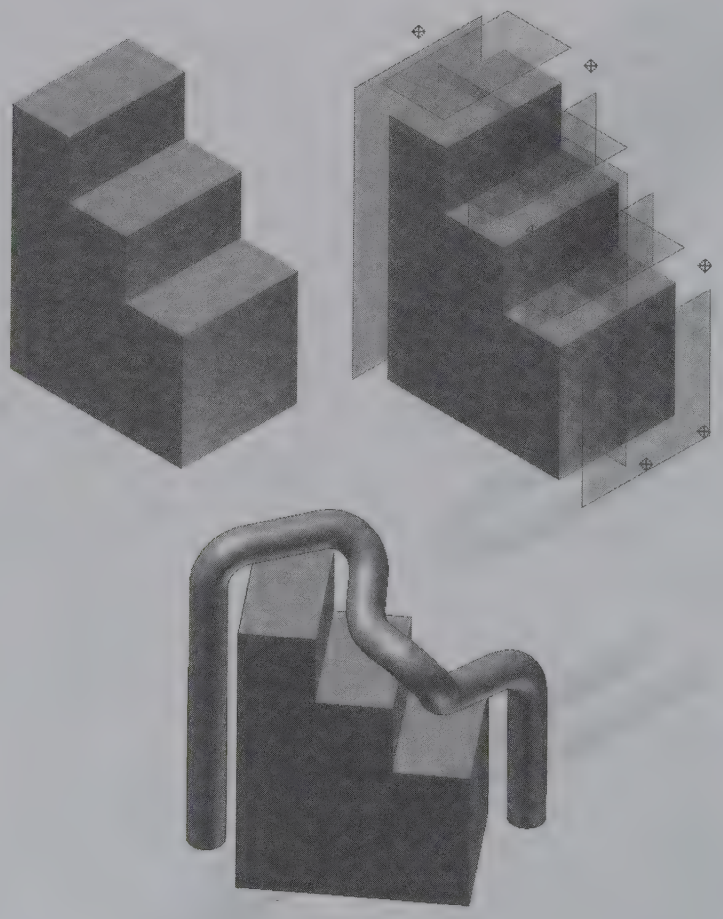
5. **Name:** Fork Body
Units: Metric
Template: PART-mm.ipt
Material: Stainless Steel, 440C
Color: As Material
Specific Instructions:
- Begin a new metric part file.
- Sketch a 4 mm × 4 mm rectangle, centered around the projected center point, on the YZ plane.
- Open a new sketch on the XY plane, and sketch the geometry shown.
- Use the **Sweep** tool to create a sweep feature using the following information:
  - **Operation Type:** Join
  - **Taper:** 0
  - **Profile:** Sketch1
  - **Path:** Sketch2
- Open a sketch on Face1, and project the edges.
- Extrude Sketch3, 150 mm in a positive direction.
- Place 2.5 mm × 15 mm chamfers of the front edges of the fork, as shown.
- Add fillets using a radius of 2 mm.
- Add rounds using a radius of .5 mm.

The final part should look like the part shown.
Save As: P9-5.ipt.



6. **Name:** Wire
Units: Inch or Metric
Template: PART-in.ipt or PART-mm.ipt
Material: Default
Color: Default
Specific Instructions:
- Begin a new inch or metric part file.
- Create an extrusion similar to the extrusion shown.
- Using the 3D sweep techniques described and the following steps, create a wire similar to the wire shown:
  - Open a 2D sketch and sketch the wire profile, a circle.
  - Finish the 2D sketch.
  - Open a 3D sketch, and offset the required work planes.
  - Use the work planes to place work points, similar to the work planes and points shown.
  - Connect the work points with a line and bends. This creates the sweep path.
  - Finish the 3D sketch.
  - Access the **Sweep** tool, and generate the sweep feature.

The final part should look like the part shown.
Save As: P9-6.ipt



7. **Name:** Casting
Units: Inch
Template: PART-in.ipt
Material: Default
Color: Default
Specific Instructions:
- Open EX9-7.ipt, and save a copy as P9-7.ipt.
- Close EX8-7.ipt without saving and open P9-7.ipt.
- Apply 2.5° of face draft to the top four split faces.
- Apply 5° of face draft to the bottom four split faces.
- Open a sketch on the top face of the casting and sketch the text shown. Use an Arial font, a text size of .24, and pick the **Middle Justification** and **Center Justification** buttons.
- Drag the corners of the text boundary to the corners of the projected face rectangle.
- Emboss the sketched text .0625", and add a Red (Flat) top face color.

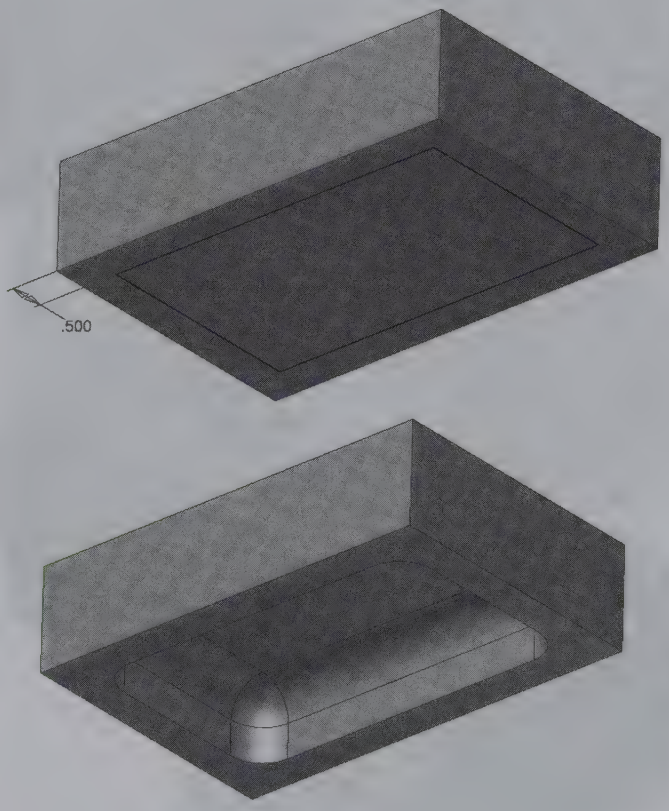
The final part should look like the part shown.
Save As: P9-7.ipt.





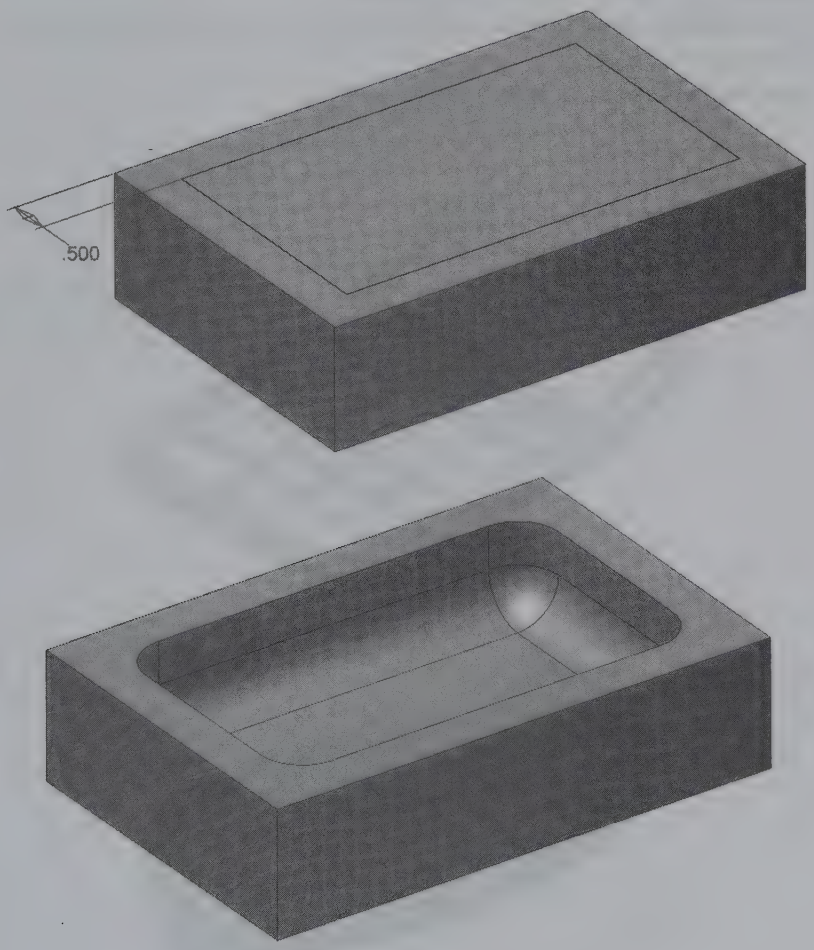
8. **Name:** Soap Pattern Top
Units: Inch
Template: PART-in.ipt
Material: Default
Color: Default
Specific Instructions:
   - Begin a new inch part file.
   - Sketch a 4″× 6″ rectangle on the XZ plane.
   - Extrude the sketch 3″ in a positive direction.
   - Offset a work plane –1.5″ from the top face of the extrusion.
   - Use the **Split** tool to split the extrusion using the following information:
     - **Method:** Split Part
     - **Split Tool:** Work Plane1
     - **Remove:** Bottom section
   - Open a new sketch on the bottom face of the split feature.
   - Offset the projected edges of the rectangle .5″ as shown.
   - Cut-extrude the sketch 1″ in a positive direction.
   - Add fillets using a radius of .5″.

   The final part should look like the part shown.
   Save As: P9-8.ipt.

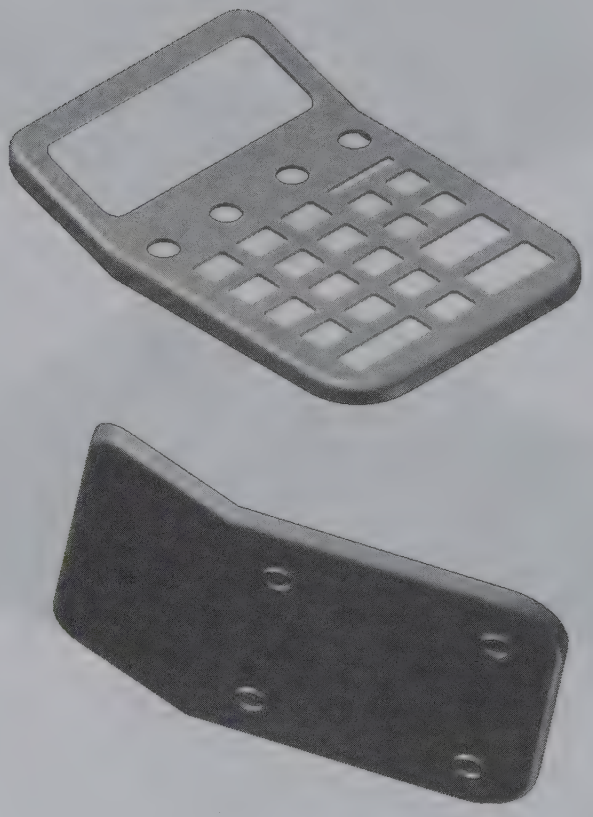


9. **Name:** Soap Pattern Bottom
Units: Inch
Template: PART-in.ipt
Material: Default
Color: Default
Specific Instructions:
- Open P9-8.ipt, and save a copy as P9-9.ipt.
- Save and close P9-8.ipt, and open P9-9.ipt.
- Delete Extrusion2 and Fillet1.
- Right-click Split1, and select the **Edit Feature** menu option.
- Pick the opposite **Side to Remove** button, and remove the top half of the extrusion.
- Open a new sketch on the top face of the split feature.
- Offset the projected edges of the rectangle .5″ as shown.
- Cut-extrude the sketch 1″ in a negative direction.
- Add fillets using a radius of .75″.

The final part should look like the part shown.
Save As: P9-9.ipt.



10. **Name:** Calculator
Units: Inch or Metric
Template: PART-in.ipt or PART-mm.ipt
Material: Default
Color: Default
Specific Instructions:
- Create a calculator face and base similar to the face and base shown.
- Generate the parts using the techniques described in this chapter and previous chapters. Although you may develop your own methods for creating the calculator parts, you may want to refer to the following list for assistance:
  - Extrude the entire calculator (face and base intact).
  - Add fillets.
  - Add work planes, or a sketch, to define the split parting line.
  - Save the part as Face.ipt.
  - Split the feature, and remove the bottom (base) section.
  - Save a copy of Face.ipt, as Base.ipt.
  - Create button slots by cut-extruding rectangles.
  - Rectangular pattern certain button slots.
  - Place holes, and add fillets.
  - Save and close the active file (Face.ipt).
  - Open Base.ipt, and edit the split to remove the top (face) section.
  - Extrude base feet.
  - Save and close the active file (Base.ipt).



11. **Name:** Decal
Units: Inch or Metric
Template: PART-in.ipt or PART-mm.ipt
Material: Default
Color: Default
Specific Instructions:

- If you have access to a .bmp, .doc, or .xls file, create a separate part decal using the information described in this chapter.

Save As: P9-11.ipt

Many complex features and techniques in this chapter are needed to create decorative products, such as this bathroom faucet. (Moen, Inc.)



CHAPTER 10

Catalog Features

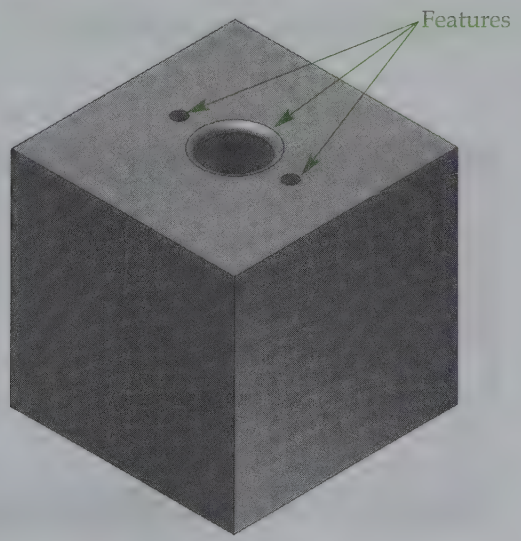
Learning Objectives

After completing this chapter, you will be able to do the following:

- View and manage iFeatures.
- Create and insert standard iFeatures.
- Edit iFeatures.
- Create and insert table-driven iFeatures.

Catalog features are similar to placed features, because you can place them on an existing part or feature. However, catalog features can be far more complex than a standard shell, fillet, chamfer, face draft, or other placed feature, and consist of several previously created features. See Figure 10-1. Catalog features, also known as ***design elements*** or ***iFeatures***, are existing features you create, such as the three holes and the fillet shown in Figure 10-1. The features are saved and stored as a single design element, in a catalog, to be used in other models. When inserted into a part, the iFeatures can retain their original size and position values, or can be modified to fit a different design application.

Figure 10-1.
An example of a simple design element (iFeature or catalog feature) and the features within it.



Viewing and Managing iFeatures

iFeatures are created as separate iFeature files (.ide) and placed in a desired location. The typical default location for iFeatures is the Catalog folder, located in the path: C:\Program Files\Autodesk\Inventor 11\Catalog, depending on your settings. This is also the location of several preexisting Inventor iFeatures. To display and manage the iFeatures located in the default Catalog folder, access the **View Catalog** tool by picking the **View Catalog** button on the **Part Features** panel bar, picking the **View Catalog** button on the **Part Features** toolbar, or picking the **View Catalog** button on the **iFeature Panel** toolbar. This opens the **Catalog** folder window in Figure 10-2. Using the specified iFeature viewer, depending on your operating system, you can navigate through folders and files to locate, view, and manage design elements. However, if you want to modify the default viewer and default design element locations, pick **Tools>Application Options...** and pick the **iFeature** tab of the **Options** dialog box. See Figure 10-3.

NOTE

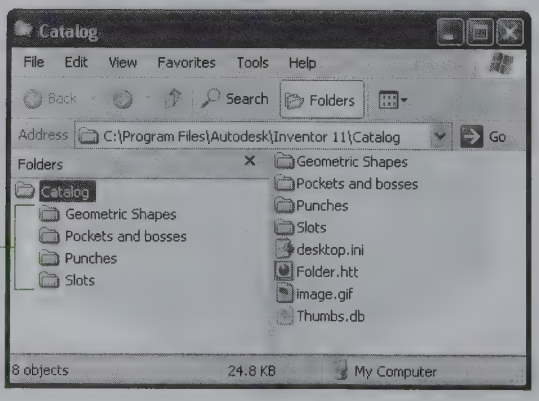


When you open a design element .ide file, very few tools are available. The purpose of opening an .ide file is to view iFeatures. You must modify iFeatures in the modeling environment.

The following options are located in the **iFeature** tab of the **Options** dialog box:

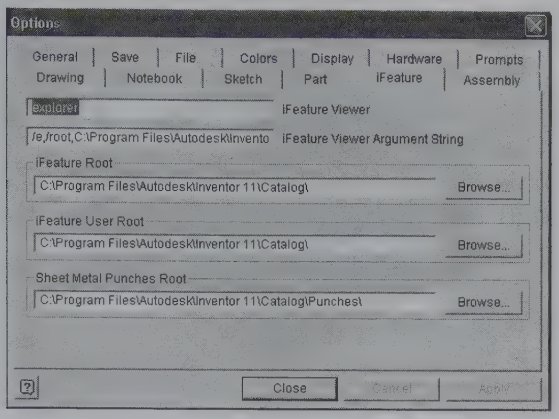
- **iFeature Viewer.** This edit box allows you to enter a viewer application for your operating system, such as Windows Explorer. The specified viewer is used to locate and manage iFeature files.
- **iFeature Viewer Argument String.** This edit box allows you to define the viewer argument string for the specified viewer. The folder listed in the edit box is automatically opened when you access the **View Catalog** tool.
- **iFeature Root.** This edit box allows you to specify the iFeature files displayed in the **View Catalog** dialog box by typing the file name in the **iFeature Root** edit box or selecting the **Browse** button and locating a different folder from the **Browse For Folder** dialog box.
- **iFeature User Root.** This edit box allows you to specify the iFeature files used by the **Create iFeature** and **Insert iFeature** dialog boxes. To change the files, type the file name in the **iFeature User Root** edit box, or select the **Browse** button and locate a different folder from the **Browse For Folder** dialog box.

Figure 10-2.
The Inventor **Catalog** folder contains iFeatures displayed using Microsoft Windows Explorer.



Existing iFeature folders

Figure 10-3.
The **iFeature** tab of the **Options** dialog box.

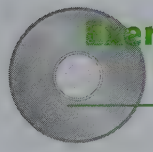


- **Sheet Metal Punches Root.** This edit box allows you to specify the iFeature files used by the sheet metal **PunchTool** dialog box by typing the file name in the **Sheet Metal Punches Root** edit box, or selecting the **Browse** button and locating a different folder from the **Browse For Folder** dialog box.

PROFESSIONAL TIP



Modify the settings in the **iFeature** tab of the **Options** dialog box to fit your needs. The default paths refer to the **Catalog** folder that is part of the Inventor program. You can change the location of your iFeature files as desired, such as placing the iFeatures in a network environment, to allow all members of the design team access to commonly used design elements.



Exercise 10-1 Complete the Exercise on the Student CD.

Creating iFeatures

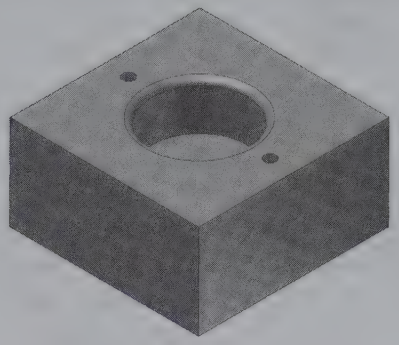
iFeatures are features that are copied from a part file and placed in an iFeature (.ide) file. You can then place the saved iFeature into another part and continually use the features you created as design elements. In order to create an iFeature, you must open an existing part file that contains features. Typically, iFeatures are developed from sketched features, such as extrusions, revolutions, holes, ribs, lofts, sweeps, and coils. You cannot create an iFeature from a placed feature, such as a shell, fillet, chamfer, or face draft. However, if a placed feature is located on and references a sketched feature, the placed feature can be associated with the design element. See Figure 10-4.

NOTE



In order to extract an iFeature of just a hole, not including the feature the hole is added to, the hole must be created using a **From Sketch** placement option.

Figure 10-4.
This iFeature consists of three holes and a fillet. The extruded block is not part of the iFeature.



Once you generate the features you want to extract into design elements, access the **Extract iFeature** tool by picking the **Extract iFeature** option from the **Tools** pull-down menu. The **Create iFeature** dialog box is displayed when you access the **Extract iFeature** tool. See Figure 10-5. The first step when creating an iFeature is to select the features that you want to use. You can pick the actual features in the graphics window, or choose the features in the browser. When you select the desired features, the features and all geometric information associated with the features are displayed in the **Selected Features** area. The first listed feature is the base feature and identifies how the iFeature is inserted into new models. Geometrically dependent features, such as a fillet on a hole edge, are also displayed. If you do not want a geometrically dependent feature to be associated with the iFeature, right-click the undesired feature and select the **Remove Feature** menu option.

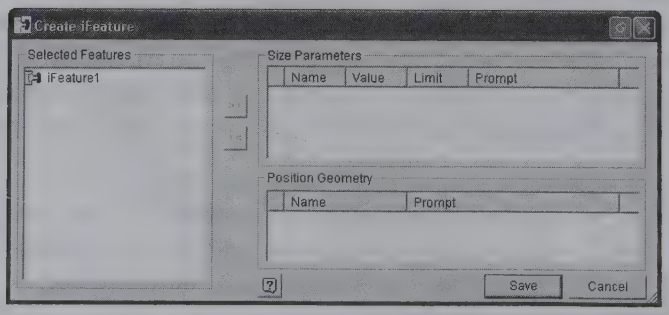
PROFESSIONAL TIP



You may want to rename the iFeature. For example, iFeature1 could receive a more descriptive name, such as Bushing Base. To rename the iFeature, select the current name, and then pick the name again or right-click and select the **Rename** option.

You can select where and how to position, insert, and size an iFeature upon insertion into a new model. However, you must set specific information when you create the iFeature to ensure the design element inserts and functions correctly. As a result, the next step in creating an iFeature is to adjust specific size parameters. As previously

Figure 10-5.
The Inventor **Create iFeature** dialog box.



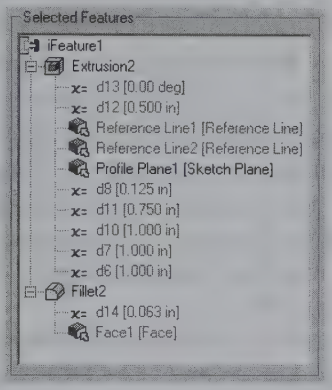
mentioned, all geometric information and parameters associated with the selected features are listed under the features in the **Selected Features** area of the **Create iFeature** dialog box. See Figure 10-6.

PROFESSIONAL TIP



The default values, such as d1, d2, and d3, indicate the order in which the parameters were created. For example, if you create an extruded cylinder, the circle diameter is d1 and the extrusion depth is d2. You may want to modify the names of the parameters in the **Parameters** dialog box so they reflect the dimension they are controlling. For example, you could change d1 to cylinder_diameter and d2 to cylinder_height.

Figure 10-6.
The **Selected Features** area of the Inventor **Create iFeature** dialog box.



When you insert an iFeature into a model, the design element geometry, or parameters can be modified if you place the parameters into the **Size Parameters** area of the **Create iFeature** dialog box by picking the parameter and selecting the **Add Parameter** button, double-clicking the parameter inside the **Selected Features** area, or right-clicking on the parameter and selecting the **Add Parameter** menu option.

Remove the parameter from the **Size Parameters** area by picking the parameter and select the **Remove Parameter** button, double-clicking the parameter for the **Selected Features** area, or right-clicking on the parameter and selecting the **Remove Parameter** menu option.

Once you insert parameters into the **Size Parameters** area of the **Create iFeature** dialog box, you can adjust the parameter name, value, limit, and prompt. See Figure 10-7. Each of these options is described as follows:

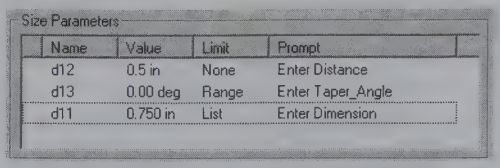
- **Name.** You may want to rename the parameter to reflect a more understandable name, such as "Major Hole Diameter." Edit the name of the parameter by selecting the default name, d12, for example, and typing a new name. When finished, pick the pencil icon or anywhere outside of the **Size Parameters** area.

NOTE



If you change the name of the parameter in the **Parameters** dialog box, the new name is displayed in the **Create iFeature** dialog box. Changing the value in the **Size Parameters** area does not modify the parameter value in the part file.

Figure 10-7.
Examples of some parameters listed in the **Size Parameters** area of the **Create iFeature** dialog box.



- **Value.** The value displayed in the **Size Parameters** area reflects the size of the parameter when it was created. Edit the parameter value by selecting the default value, 25 mm for example, and typing a new value. When finished, pick the pencil icon or anywhere outside of the **Size Parameters** area.
- **Limit.** This option allows you to place limits on the amount of size changes that can occur when an iFeature is inserted into a part. You can specify no limit by selecting the **None** option, which is the default. You can also specify a **Range** or **List** limit. These three options are found in a **Limit** drop-down list. To create a limited size range, select the **Limit** drop-down list, and choose the **Range** option. Selecting **Range** opens the **Specify Range** dialog box for the selected parameter. See Figure 10-8. The **Specify Range** dialog box allows you to define a minimum limit, less than a specified value, less than or equal to a specified value, or infinitely less than the default value. You can also define a maximum limit, greater than a specified value, greater than or equal to a specified value, or infinitely greater than the default value. Finally you can modify the default value in the **Default** edit box, which is the same as changing the **Value**, previously described.

  To create a limited size list, select the **Limit** drop-down list, and choose the **List** option. Selecting **List** opens the **List Values** dialog box for the selected parameter. See Figure 10-9. The **List Values** dialog box allows you to define a certain number of values. The values you specify in this edit box are available for selection when the design element is inserted. The current default value is automatically listed in the **List** dialog box. To add another optional value, select the **Click here to add value** button and define the value. You can then choose the new value from the **Default** drop-down list, which is the same as changing the **Value**, previously described.
- **Prompt.** When an iFeature is inserted, a prompt instructs you on the parameter value operation, such as "**Enter distance.**" The **Prompt** option allows you to

Figure 10-8.
The **Specify Range** dialog box.

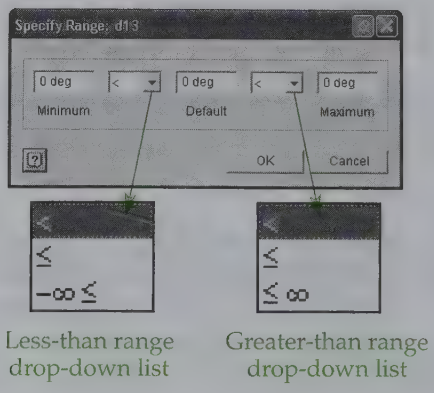
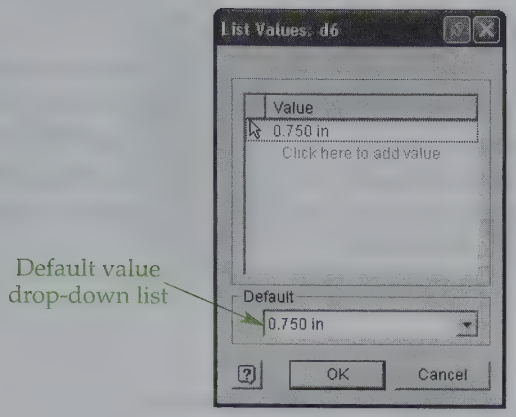


Figure 10-9.
The **List Values** dialog box.



modify the default prompt to a more understandable description. For example, you may want to change the default hole depth prompt of "Enter distance," to "Enter the hole depth."

NOTE

In addition to physical feature parameters, reference geometry parameters are also listed in the **Selected Features** area of the **Create iFeature** dialog box. These parameters are identified by a reference icon.

The next step to creating an iFeature is to specify how you want to position the iFeature geometry when the iFeature is inserted, using the **Position Geometry** area of the **Create iFeature** dialog box. When you insert the iFeature, the feature position can only reference the additional reference parameters you place into the **Position Geometry** area of the **Create iFeature** dialog box. For example, if the only position geometry listed in the **Position Geometry** area is Profile Plane1 you are only able to select, and reference, a profile plane of the part on which the iFeature is inserted. To add additional position geometry to the **Position Geometry** list box, such as a reference edge (Reference Line1), pick the position geometry and select the **Add Geometry** button, double-click the position geometry inside the **Selected Features** area, or right-click the position geometry and select the **Expose Geometry** menu option.

Remove position geometry from the **Position Geometry** area by picking the position geometry and selecting the **Remove Geometry** button, double-click the position geometry inside the **Selected Features** area, or right-clicking the position geometry and selecting the **Remove Geometry** menu option.

Once you insert the position geometry into the **Position Geometry** area of the **Create iFeature** dialog box, you can adjust the position geometry name and prompt the same way you adjust the name and prompt of the parameters previously described. In addition, when you right-click certain position geometry listed in the **Position Geometry** area of the **Create iFeature** dialog box the **Make Independent** menu option is available, which allows you to divide shared position geometry. For example, if an extrusion and a fillet share the same position geometry, you are able to make the extrusion and fillet position geometry independent of each other. The **Make Independent** option is only available when two or more features are selected. The **Combine Geometry** menu option is also available, which is opposite of the **Make Independent** option, and allows you to combine shared position geometry. After you choose the **Combine Geometry** option, select the additional geometry you want to combine with the selected objects.

Once you specify all the options in the **Create iFeature** dialog box, select the **Save** button to access the **Save As** dialog box. Saving an iFeature is similar to saving other files. You may notice how the default save location corresponds to the **iFeature User Root** specified in the **iFeature** tab of the **Options** dialog box.

NOTE

For the most efficient use of iFeatures, you may want to adjust the options located in the **iFeature** tab of the **Options** dialog box and create an iFeature library or set of libraries. You can then save your design elements in these specific locations.

Exercise 10-2

Complete the Exercise on the Student CD.

Inserting iFeatures

Once you create an iFeature, you are ready to insert it into a part using the **Insert iFeature** tool by picking the **Insert iFeature** button on the **Part Features** panel bar, or picking the **Insert iFeature** button on the **Part Features** toolbar. This opens the **Insert iFeature** dialog box in Figure 10-10. The first step when inserting an iFeature is to pick the **Browse...** button to access the **Open** dialog box. The **Open** dialog box allows you to locate an iFeature anywhere on your computer or the network. Once you select an iFeature, the **Position** window is activated and a preview of the iFeature is displayed in the graphics window. See Figure 10-11. The prompt in the **Position** window of the **Insert iFeature** dialog box guides you through the process of positioning the iFeature. Typically, you need to pick the face of an existing feature or the work feature where you want to place the iFeature. You then have the option of rotating the iFeature, by selecting the default angle in the list box, and entering a new angle, or by picking the **Rotate** icon and rotating the iFeature. Similarly, you can move the iFeature by selecting the **Move** icon and dragging the iFeature to the desired location.

After you position the iFeature, pick the **Next** button, or select the **Size** option on the tree structure to activate the **Size** window. See Figure 10-12. If you entered specific

Figure 10-10.
The Inventor **Insert iFeature** dialog box, **Select** window.

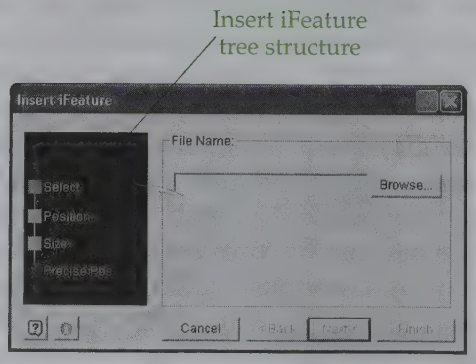
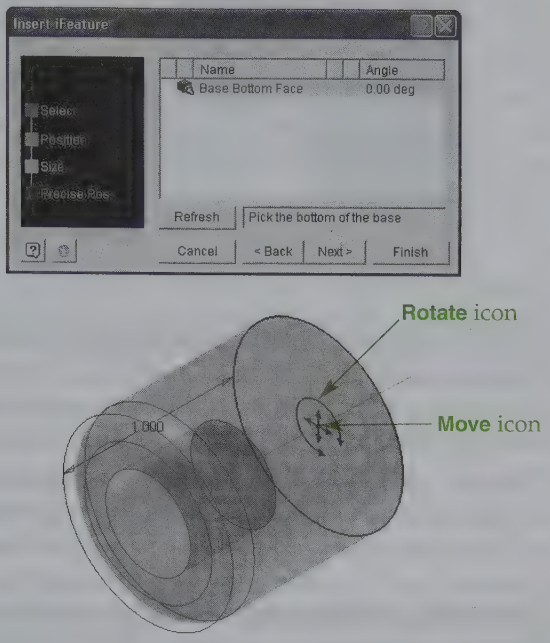


Figure 10-11.
The Inventor **Insert iFeature** dialog box, **Position** window is open while the **Rotate** and **Move** icons are visible on the inserted iFeature.



size parameter information when the iFeature was created, the size parameters are listed in the **Size** list box. The **Size** window prompts guide you through the iFeature sizing process. If no size limits were placed on the iFeature when it was created, you can enter any value. However, if a range limit was placed on the iFeature when created, you will only be able to enter a specific range of values, or if a list limit was placed on the iFeature when created, you will only be able to choose values from the **Value** drop-down list.

NOTE

To preview the changes, pick the **Refresh** button in the **Insert iFeature** dialog box any time you resize or reposition the iFeature.

Once you make size modifications to the iFeature, pick the **Next** button, or select the **Precise Pos.** option on the tree structure to activate the **Precise Position** window. See Figure 10-13. This window contains two radio buttons, which allow you to specify whether or not you want to further position the iFeature with sketch constraints

Figure 10-12.
The **Insert iFeature** dialog box, **Size** window.

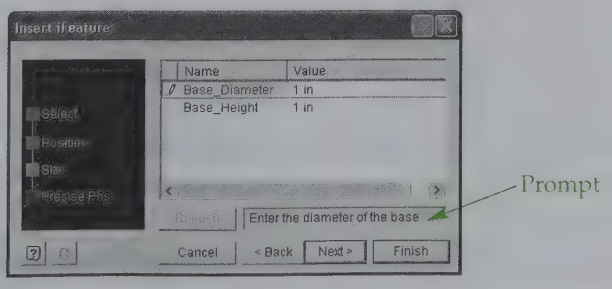
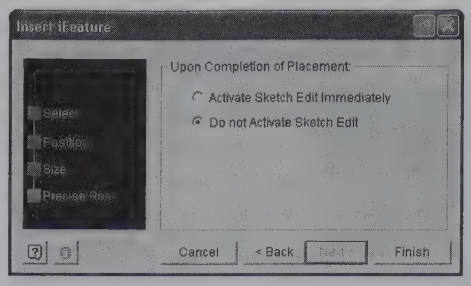


Figure 10-13.
The **Insert iFeature** dialog box, **Precise Position** window.



and dimensions. When the **Activate Sketch Edit Immediately** radio button is selected, the iFeature sketch is automatically opened, allowing you to precisely position the iFeature features using sketch constraints and dimensions. When the **Do Not Activate Sketch Edit** radio button is selected, the iFeature sketch does not automatically open. Consequently, to precisely position the iFeature, using constraint and dimension sketching tools, you must open the iFeature sketch by editing.

Once you select, position, and size the iFeature you want to insert and specify the desired precise position option, the final step to insert an iFeature is to select the **Finish** button. However, if you at least select and position the iFeature, you can pick the **Finish** button and bypass the other insertion options.

NOTE



Pick the **Back** button in the **Insert iFeature** dialog box any time you want to return to the previous window to modify an insertion option. You can also choose any of the options on the tree structure and return to modify an insertion option.



Complete the Exercise on the Student CD.

Editing iFeatures

iFeatures can be edited to explore design alternatives and product changes. You can redefine iFeatures in the part environment and in the iFeature work environment depending on the required changes. While in the part environment, use the following iFeature modification options:

- **Edit the iFeature sketch.** Whenever you insert an iFeature, a sketch is created that defines the precise location of the iFeature. You can open this sketch when the iFeature is inserted, or you can open the sketch any time to modify the location of the features. To edit an iFeature sketch, right-click on an iFeature or an iFeature sketch in the browser, and pick the **Edit Sketch** pop-up menu option.
- **Edit the iFeature.** If you want to redefine the iFeature position and size properties specified when you inserted the iFeature, edit the iFeature by right-clicking on an iFeature or an iFeature sketch in the browser, and picking the **Edit iFeature...** pop-up menu option.

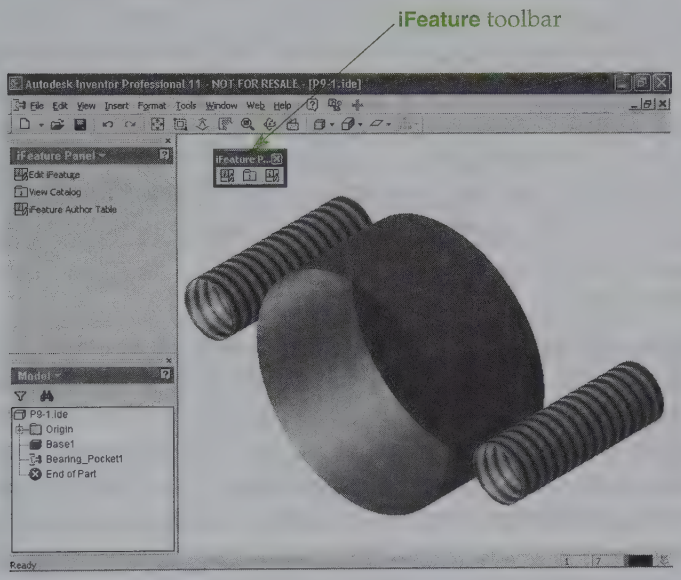
You can also work in the iFeature environment by opening an iFeature (.ide) file. See Figure 10-14. An .ide file contains specific iFeature definition tools, the iFeature, and related items such as a solid and sketch, if applicable. In addition to the **View Catalog** tool, the **Edit iFeature**, and **iFeature Author Table** tools are available inside an iFeature file. The **iFeature Author Table** allows you to generate table-driven iFeatures. The **Edit iFeature** tool is available by selecting the **Edit iFeature** button from either the **iFeature** panel or the **iFeature** toolbar. The **Edit iFeature** tool opens the **Edit iFeature** dialog box, which is identical to the **Create iFeature** dialog box. Use the **Edit iFeature** dialog box in the iFeature work environment to redefine the options you specified when you extracted the iFeature.

NOTE



While in the iFeature work environment, you can redefine the display characteristics of the iFeature icon by selecting the **Change Icon...** option from the **File** pull-down menu.

Figure 10-14.
An example of an opened iFeature (.ide) file work environment.

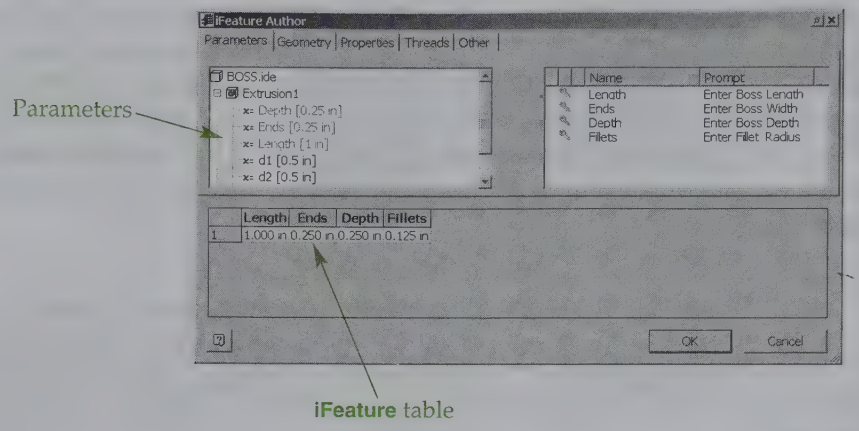


Creating Table-Driven iFeatures

When you create an iFeature, you can specify which size and position parameters can be modified when the iFeature is inserted into a model. For most applications, this approach to iFeature creation and insertion is effective, allowing you to insert an iFeature and change size and position parameters as needed. However, another option for creating and inserting iFeatures is to use a table. A ***table-driven iFeature*** allows you to create multiple variations of the original iFeature, using information stored in a spreadsheet. As a result, you can place numerous different iFeatures that are based on the original iFeature, without having to generate additional parts or iFeature files. Each time you insert a table-driven iFeature, you can select from specific size and shape characteristics that you define when the iFeature table is created.

You can create a table-driven iFeature by selecting the **iFeature Author Table** button from either the **iFeature Panel** bar or the **iFeature** toolbar from inside an iFeature (.ide) file. Selecting the **iFeature Author Table** tool opens the **iFeature Author** dialog box.

Figure 10-15.
The **iFeature Author** dialog box, **Parameters** tab.



See Figure 10-15. The **iFeature Author** dialog box has tabs and options that allow you to fully define a table-driven iFeature. In general, the **iFeature Author** dialog box allows you to identify which model parameters, geometry, properties, threads, and custom values you want to place in the iFeature spreadsheet and have available when inserting the iFeature. Each of these preferences is specified using the options in the corresponding tab. As you define iFeature information using the tabs, columns that relate to the tab items are created in the **iFeature Table**, in addition to a single row, which contains column values, or cells. Then you can add rows to the **iFeature Table** to define additional iFeature values, allowing you to select and create different iFeature instances.

The Parameters tab

The **Parameters** tab, shown in Figure 10-15, displays specified iFeature size and shape parameters. As you can see, two lists are available. The **Parameters** list on the left side of the dialog box displays the iFeature and all the features you selected when creating the iFeature. Listed under each feature are the corresponding parameters that define the feature size and shape. The **Selected Items** list on the right side of the dialog box identifies which parameters are used to define the iFeature when placed in a part. The selected items represent the parameter columns in the **iFeature Table**, which are the features that you can manipulate when the iFeature is inserted into a part. Before creating a table-driven iFeature, you must name parameters, because only parameters that have been named using the **Parameters** or **Dimension Properties** dialog box while in the part file, or the **Size Parameters** area of the **Create iFeature** dialog box, are displayed in the **Selected Items** list.

Notice that there are key symbols displayed next to each of the selected parameters. ***Keys*** identify which parameters, geometry, properties, threads, and custom items you want to use to define the part instance when you place an iFeature. For example, when working with parameters, if you want to create a ∅1" (24.5 mm) hole feature, with multiple depths to select from, you can key the depth parameter. You can specify nine keys in the entire table-driven iFeature, which means that nine keys can be spread throughout the five different tabs. Each tab does not have nine separate keys. Keys are listed in order when you insert the table-driven iFeature into a part. Key 1 is known as the primary key, while keys 2 through 9 are secondary keys. To identify the primary key in the parameter tab, pick the key symbol or right-click on the parameter in the **Selected Items** list, and select key 1 from the **Key** cascading submenu. If you select another parameter key symbol, key 2 is defined. Additional keys can also be specified by right-clicking on the parameter in the **Selected Items** list and selecting the desired key from the **Key** cascading

submenu. Selected keys are blue and have a key number next to them. Remove a key by selecting the key symbol a second time, or by right-clicking on the parameter in the **Selected Items** list, and selecting the **Not A Key** option from the **Key** cascading submenu.

NOTE

If you do not specify any keys, all parameters are listed when you insert a table-driven iFeature.

The Geometry tab

The **Geometry** tab, shown in Figure 10-16, contains the position geometry you specified when you extracted the iFeature. Use this tab to change the position geometry name and prompt or add keys as described in the **Parameters** tab.

The Properties tab

The **Properties** tab, shown in Figure 10-17, displays and allows you to select model feature properties identified in the **Properties** dialog box of the iFeature file. As with the **Parameters** tab, two lists are available in the **Properties** tab. The **Properties** list on the left side of the dialog box displays the iFeature and all the iFeature properties available and defined in the **Properties** dialog box. Listed under each property type are all the related properties. The **Selected Items** list on the right side of the dialog box identifies which properties you select to include in the iFeature spreadsheet. Again, the selected items are typically those properties you want to manipulate when the iFeature is inserted. For example, if you want to add an iFeature description, you must ensure the physical property controlling the iFeature description is listed in the **Selected Items** list.

NOTE

Even if a property value does not vary between instances, such as the part number, you must include the property in the **Selected Items** list if you want the property to be available in bills of material, and drawings.

To add a property from the **Properties** list to the **Selected Items** list, select the property you want to add from the **Properties** list and pick the **Add** button, or double-click on the item. The name of the property in the **Selected Items** list is defined in the **Properties**

Figure 10-16.
The **iFeature Author** dialog box, **Geometry** tab.

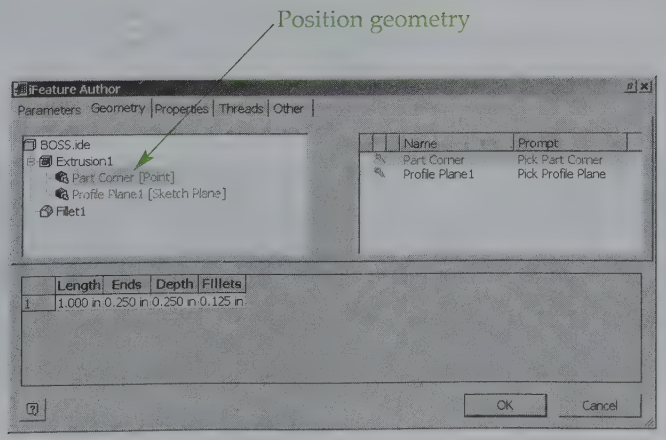
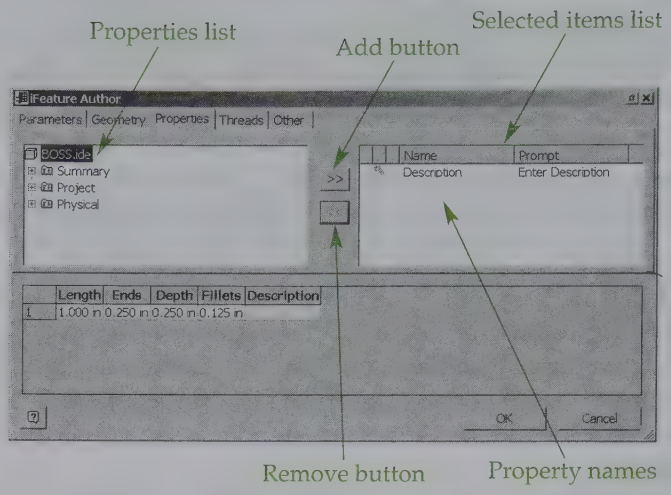


Figure 10-17.
The **iFeature Author** dialog box, **Properties** tab.



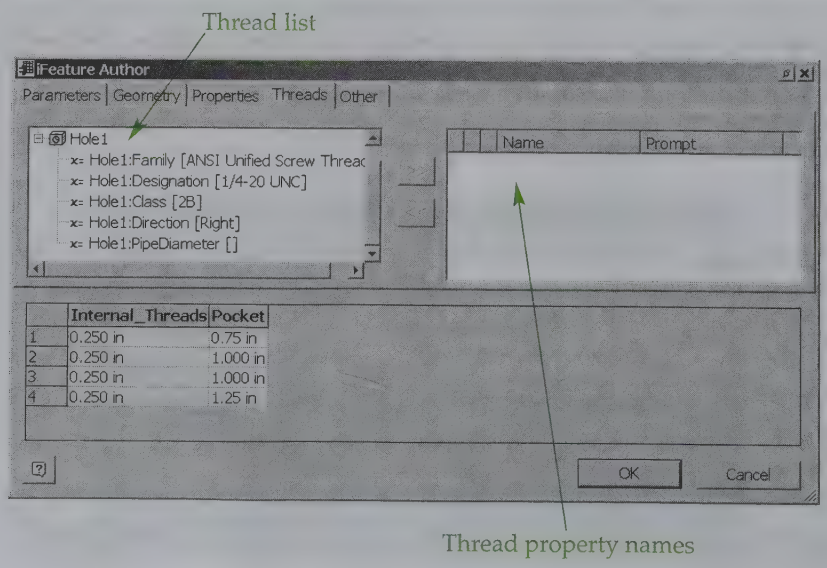
dialog box, and corresponds to the column header in the **iFeature Table**, as you can see by the **Description** column in Figure 10-17. If the property has a value such as Boss for a description property, the specified value is listed in the table cell or cells. To remove a property in the **Selected Items** list, select the property and then pick the **Remove** button or right-click on the property and select the **Delete Column** menu option.

There are key symbols displayed next to each of the properties that allow you to identify which properties are available throughout the design process. If you do not specify any keys, all properties are listed when you insert an iFeature.

The Threads tab

The **Threads** tab, shown in Figure 10-18, displays and allows you to select thread specifications for iFeatures that contain threaded features. As with the **Parameters** tab, two lists are available in the **Threads** tab. The **Threads** list on the left side of the dialog

Figure 10-18.
The **iFeature Author** dialog box, **Threads** tab.



box displays the iFeature and all the features that contain threads. Listed under each feature are the thread variables that correspond to the threaded feature parameters. The **Selected Items** list on the right side of the dialog box identifies what thread information you select to include in the iFeature spreadsheet and have available when the iFeature is inserted. The name and prompts correspond to the values defined in the threads spreadsheet file (Threads.xls).

To add a thread variable from the **Threads** list to the **Selected Items** list, select the item you want to add and pick the **Add** button, or double-click on the item. The specified value is automatically keyed and listed in the **iFeature Table**. To remove a property in the **Selected Items** list, select the property, and then pick the **Remove** button or right-click on the property and select the **Delete Column** menu option.

There are key symbols displayed next to each thread characteristic that allow you to identify which thread items are available throughout the design process. If you do not specify any keys, all thread items are listed when you insert an iFeature.

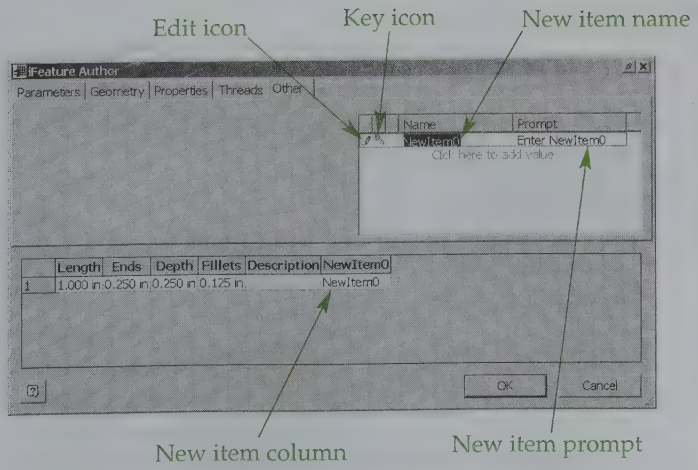
The Other tab

The **Other** tab, shown in Figure 10-19, displays and allows you to create additional iFeature placement options. Most iFeature attributes are controlled by the specifications in the **Parameters**, **Geometry**, **Properties**, and **Threads** tabs of the **iFeature Author** dialog box. Any other custom attributes that cannot be defined in these four tabs can be created in the **Other** tab. For example, maybe you want to define additional iFeature placement information or characteristics. To create a custom item, pick the **Click here to add value** button. As shown in Figure 10-19, default "NewItem0" name and "Enter NewItem0" prompts are added to the **Item** list, and a corresponding column with the same name is created. To rename the item or change the prompt, ensure the **Edit** icon is displayed by selecting the current item name or prompt. Then, enter the name of the custom attribute or a new prompt, followed by selecting the **Edit** symbol to apply the changes. You can remove a custom item by right-clicking on the item and selecting the **Delete Column** menu option.

The iFeature factory table

Once you fully identify all the items you want to manipulate when inserting an iFeature, you are ready to work with the iFeature factory table. In many cases, only one row is initially created. This row is based on the options you specify in each of the **iFeature Author** dialog box tabs. You can modify the values in each of the cells to reflect the desired iFeature attributes. Still, a single row means that only one instance of the iFeature can be inserted

Figure 10-19.
The **iFeature Author** dialog box, **Other** tab.



into a part. For example, if you create an iFeature of a Ø2" (49 mm) hole, and define only one row with no value range, you are only able to insert a Ø2" (49 mm) hole into a part, which is not the purpose of using table-driven iFeatures. Table-driven iFeatures are used to insert multiple types, or instances, of features. Adding additional rows to the **iFeature Table** allows you to define these multiple iFeature instances.

NOTE

If you do not specify a unit of measurement in a table cell, the default document units are applied.

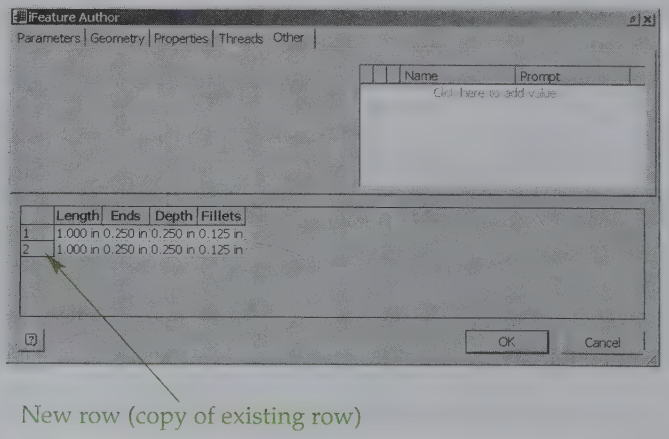
To create another **iFeature Table** row, right-click on the existing row and select the **Insert Row** menu option. A copy of the row you select with all the existing cell values is added to the **iFeature Table**. See Figure 10-20. Just as with the initial table row, you can modify the values in each of the cells to reflect the desired iFeature instance attributes. For example, if you want one extrusion length option to be 6 in. (147 mm), and another extrusion length option to be 8 in. (203 mm), enter 6 in. in the length cell of the first row, and 8 in. in the length cell of the second row. You can enter any value you want in the table cells, but suppression values require certain expressions. To specify a feature as suppressed, you can enter Suppress, S, s, OFF, Off, off, or O. To identify a feature as computed, you can enter Compute, U, u, C, c, ON, On, on, or 1. Continue adding rows and defining cell values as required, generating the desired number of iFeature instances. You can also delete a cell row by right-clicking on the row you want to remove and selecting the **Delete Row** menu option.

NOTE

To define a row as the default instance of the iFeature for inserting into a part, right-click and select the **Set as Default Row** menu option.

When you specify a certain value in a cell, such as 3 or 5, these values are considered standard and cannot be modified when an iFeature is inserted into a part. However, you can identify custom parameters that can be defined as any value, or a specific range of values, when the iFeature is inserted. You can establish a custom column in which all cell values can be modified, or a custom cell in which only the specified cell value can be modified. To create a custom column, right-click on the column header in the **iFeature Table** or the column name in the **Selected Items** list.

Figure 10-20.
Creating an additional iFeature table row.



Then, pick the **Custom Parameter Column** menu option. Now when you insert the iFeature into a part, you can specify any value for the parameter, and create any size feature based on the column information. Another option is to specify a certain range of values that can be used to build the iFeature. To identify a range, right-click on the column header in the **iFeature Table** or the column name in the **Selected Items** list that has been established as a custom parameter column. Then pick the **Specify Range For Column...** menu option to display the **Specify Range** dialog box.

NOTE



You cannot specify a column or cell range for certain items including some properties, position geometry, and threads. Custom columns and cells display blue backgrounds.

Working with iFeature Factories

Once you fully define all the preferences in the **iFeature Author** dialog box, pick the **OK** button to generate the iFeature attributes. The final step in creating a table-driven iFeature factory is to resave the iFeature file. Often iFeatures are used in multiple parts, which means you should save the iFeature in the active project.

There are a number of browser options related to working with table-driven iFeatures once they are created. See Figure 10-21. Notice the iFeature file name is listed first, followed by the base solid feature, and then the iFeature table. The iFeature table contains all the information you specify using the **iFeature Author** dialog box. For example, the iFeature table shown in Figure 10-21 has two rows. Each row contains the various columns you define in the **iFeature Author** dialog box tabs. Only keyed attributes are listed in the table. The check boxes next to column items indicate the iFeature currently displayed in the graphics window of the iFeature file. To observe a different iFeature instance, double-click on the last attribute. For example, in Figure 10-23 double-click "Fillets = 0.25 in." to observe the 2 in. iFeature, or double-click "Fillets = 0.125 in." to observe the 1 in. iFeature.

There are two options for editing iFeature tables. One method is to right-click on the iFeature table in the browser and select the **Edit Table...** menu option, or double-click the iFeature table. Using these techniques reopens the **iFeature Author** dialog box, allowing you to manipulate the initial table-driven iFeature specifications. Another editing method is to right-click on the iFeature table in the browser and select the **Edit via Spread Sheet...** menu option to access the embedded Microsoft Excel® spreadsheet, as shown in Figure 10-22.

Figure 10-21.
An example of the browser display for a table-driven iFeature.

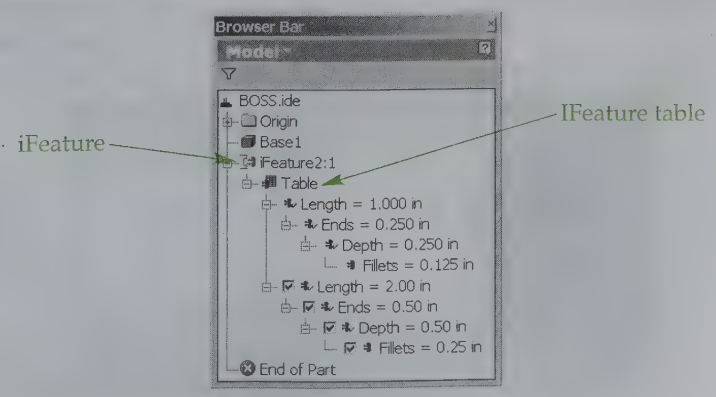
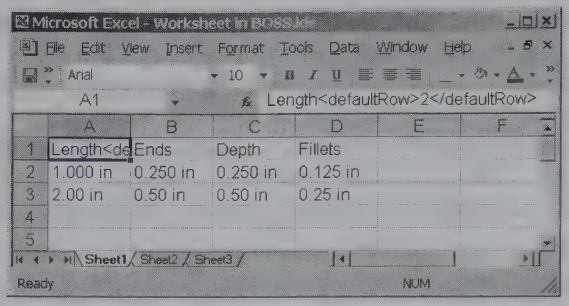


Figure 10-22.
An example of an embedded Microsoft Excel® spreadsheet created when you generate a table-driven iFeature.



NOTE

For more information regarding spreadsheets, you may want to access the Autodesk Inventor or Microsoft Excel help files.

Exercise 10-4

Complete the Exercise on the Student CD.

Inserting Table Driven iFeatures

Once you create a table driven iFeature, use the **Insert iFeature** tool to access the **Insert iFeature** dialog box, and place the iFeature into a part file. Table-driven iFeatures are identified in the **Insert iFeature** dialog box by the **Table Driven** icon shown in Figure 10-23. The process of selecting and positioning a table-driven iFeature is the same process used for placing a standard iFeature as described earlier in this chapter. However, as shown in Figure 10-23, when you size the iFeature, the specified table options are available.

Custom attributes are not keyed and can be edited like any standard iFeature size attribute by entering or selecting a new value. In contrast, when you select the default value of a keyed item in the **Value** list, the **Select Value** list box is displayed. See Figure 10-24. To

Figure 10-23.
The **Insert iFeature** dialog box, **Table Driven** iFeature.

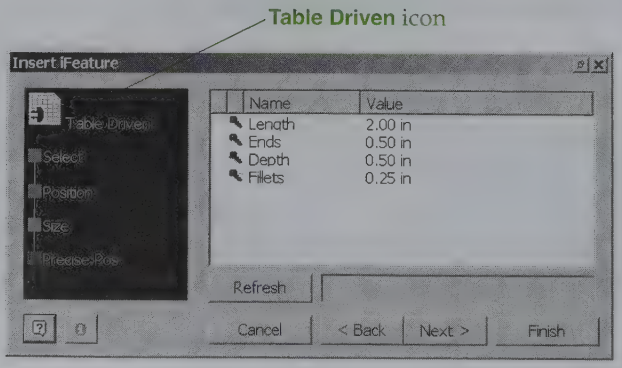


Figure 10-24.
The **Select Value** list box.



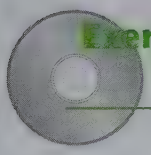
show all the specified table cell values, pick the **All Values** check box. When you select a value, the **Select Value** list box closes and the new value is displayed in the **Value** list. To exit the **Select Value** list box without choosing a value, press the [Esc] key on your keyboard. Then, continue defining the iFeature instance by selecting values until you fully establish the iFeature parameters.

Just as when inserting standard iFeatures, once you make size modifications to the table driven iFeature, pick the **Next** button, or select the **Precise Pos.** option on the tree structure to activate the **Precise Position** window. Specify whether you want to further position the iFeature with sketch constraints and dimensions, and then select the **Finish** button to complete the table driven iFeature insertion.

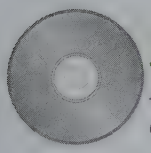
NOTE



Edit table driven iFeatures using the same standard iFeature editing techniques previously described.



Exercise 10-5 Complete the Exercise on the Student CD.



Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. Give another name for catalog features.
2. Describe *catalog features.*
3. How are catalog features far more complex than a standard shell, fillet, chamfer, face draft, or other placed features?
4. Where is the typical default location for iFeatures?
5. What is a *table driven iFeature* and how does it work?
6. Describe and give an example of *keys.*
7. How many keys can you specify in the entire table driven iFeature?
8. How are keys listed when you insert the table driven iFeature into a part?
9. Identify the *primary key* and the *secondary keys.*
10. Discuss the disadvantage of having only a single row in the iFeature table.

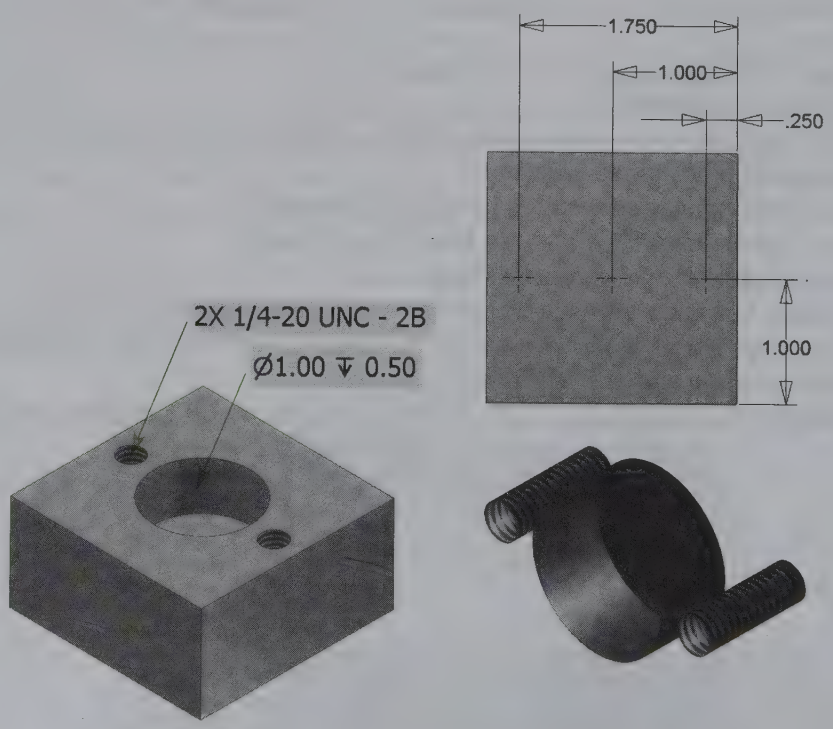
Problems

Instructions:

- Follow the specific instructions provided for each problem.

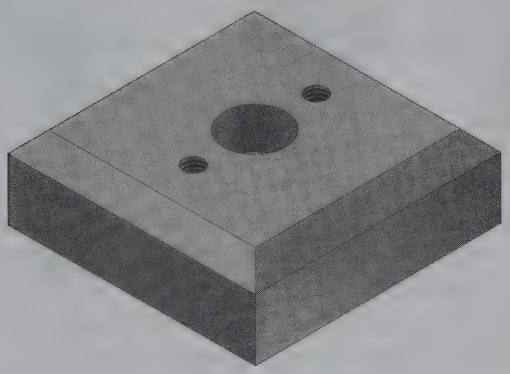
1. **Name:** Bearing Pocket
   Units: Inch
   Template: PART-in.ipt
   Material: Default
   Color: Default
   Specific Instructions:
   - Begin a new inch part file.
   - Create a 1″ × 2″ × 2″ extruded rectangle.
   - Open a sketch on the top face of the extrusion, and place the hole centers as shown.
   - Create the two smaller holes as specified.
   - Share Sketch2.
   - Create the larger middle hole as specified.
   - Use the **Parameters** dialog box to rename the internal thread diameter (Hole1) to Internal_Threads, and the pocket diameter (Hole2) to Pocket.
   - **Create an iFeature using the following information:**
     - Name: Bearing_Pocket
     - Selected Features: Hole1 and Hole2.
     - Limit the value of "Internal_Threads" to a range of .125 < .25 < .5.
     - Change the prompt of "Internal_Threads" to "Enter Diameter of Internal Threads."
     - Limit the value of Pocket to a list of .75″, 1″, and 1.25″.
     - Change the prompt of "Pocket" to "Enter Pocket Diameter."
     - Save Bearing_Pocket as P10-1.ide.
   - Access the **View Catalog** tool, and open P10-1.ipt.

   The iFeature should look like the feature shown.
   Save As: P10-1.ipt.

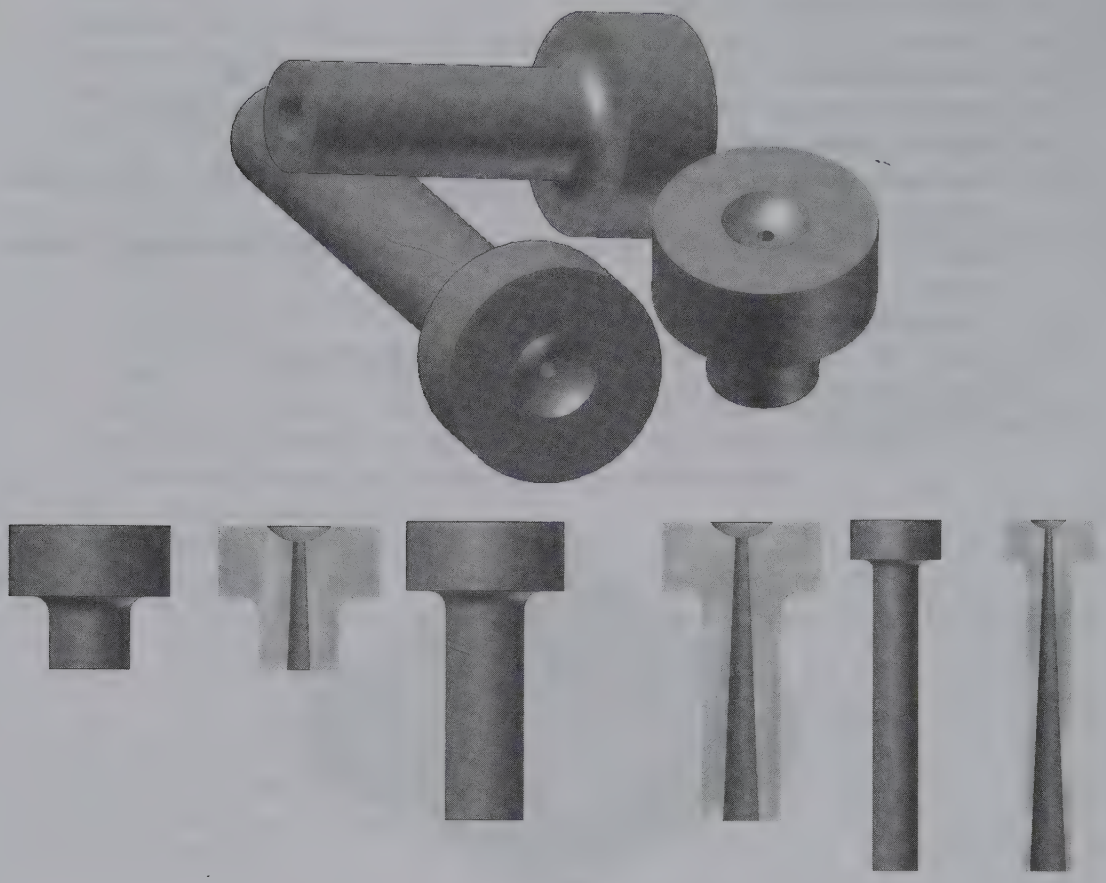


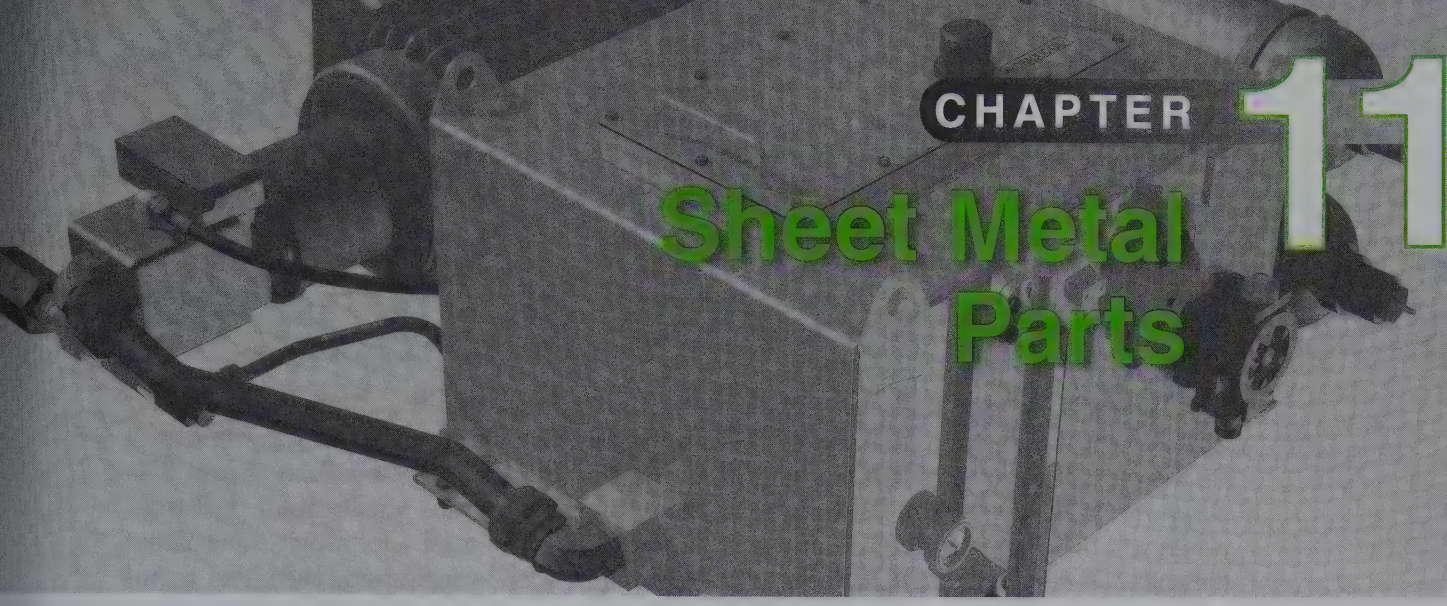
2. **Name:** Bearing Pocket Base
Units: Inch
Template: PART-in.ipt
Material: Default
Color: Default
Specific Instructions:
   - Begin a new inch part file.
   - Create a 1″ × 3″ × 3″ extruded rectangle.
   - Insert the iFeature, P10-1.ide, on the top face of the extrusion.
   - Attempt to enter a value of .5″ in the **Internal_Threads Value** edit box.
   - Select the **Activate Sketch Edit Immediately** radio button.
   - Use sketch dimensions, center the iFeature in the middle of the extrusion, and finish the sketch.
   - Right-click Bearing_Pocket1 in the browser and select the **Edit iFeature...** menu option.
   - Change the bearing pocket diameter to .75″.
   - Place a .25″ chamfer around the top edge of the extrusion.

   The final part should look like the part shown.
   Save As: P10-2.ipt.



These sprue bushings are used in injection molds. The inside shape, shown in the sectioned views, is the same. The outside shape can, and often is, designed for a single mold. You can save the features that create the inside shape as an iFeature. When a new sprue bushing is needed, create the outside shape and add the iFeature to create the inside shape. This method will ensure consistency.





CHAPTER 11

Sheet Metal Parts

Learning Objectives

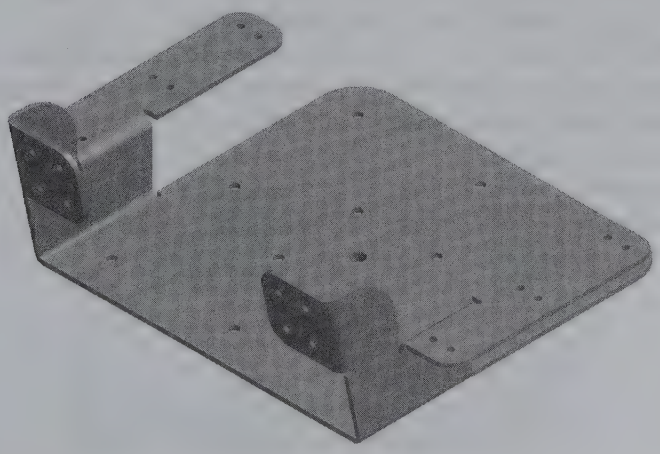
After completing this chapter, you will be able to do the following:

- Use and define sheet metal styles.
- Create sheet metal sketched features.
- Add sheet metal placed features.
- Use the sheet metal PunchTool.
- Create flat patterns.

Sheet metal parts are created by forming flat metal into a particular shape. Typically, a sheet metal drawing consists of a flat pattern, which represents the final, unfolded part. The sheet metal pattern is transferred to a flat sheet of metal, which is then sheared and bent to the correct specifications. Sheet metal drafting and sheet metal parts are used in a number of industries, including heating, ventilating, and air conditioning (HVAC), automotive, and product chassis. See Figure 11-1.

The process of creating sheet metal parts is very similar to the process of creating other parts. Usually, you begin with a sketched feature and then add additional features to the part. However, Inventor provides you with specific tools that allow you

Figure 11-1.
An example of an Inventor sheet metal part.



to develop sheet metal parts. For most sheet metal applications, the metal thickness, material, bend radii, and relief sizes remain the same throughout part construction. As a result, you should define sheet metal parameters first, so the style characteristics are applied to the part while you develop your design.

Sheet Metal Setup and Format

Typically, when you begin a new design and open a new part file, you specify whether the part is for a modeling application or a sheet metal application, depending on the type of part you want to create. Both part model files and sheet metal part files carry the Inventor part file extension of .ipt. However, specific sheet metal part templates are available by default, and are listed as Sheet Metal.ipt. Opening a new Sheet Metal.ipt file template creates a part as a sheet metal subfile and opens the sheet metal tools necessary to create sheet metal parts when required.

After you create a sketch, enter the sheet metal feature work environment by right-clicking and selecting the **Finish Sketch** option from the pop-up menu; picking the available sheet metal tool from the **Sheet Metal Features** panel bar or **Sheet Metal Features** toolbar; selecting the **Return** button on the **Inventor Standard** toolbar; or pressing the [S].

When beginning a new sheet metal design, you should use a sheet metal template. However, if you begin a new sheet metal part in a modeling application and want to access sheet metal tools, pick the **Sheet Metal** option from the **Applications** pull-down menu. Selecting this menu option allows you to change from a **Modeling** design to a **Sheet Metal** design while in the part model work environment.

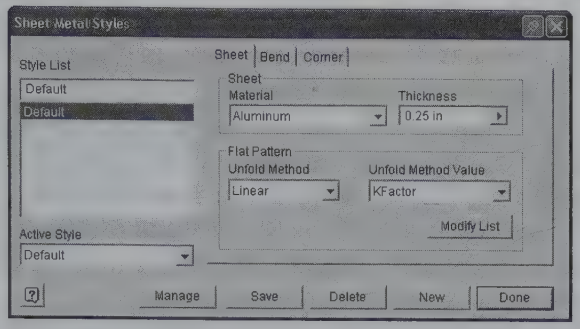
Sheet Metal Styles

Several sheet metal parameters used when creating sheet metal parts are specified before you begin the design. These parameters are automatically applied to the design as you create the part. Though you can define the bend radius, unfold options, and bend relief options while you create a sheet metal part, you should specify these options and all other parameters, such as the material type, thickness, and corner relief options, before you begin a design. This technique replicates real-world sheet metal part manufacturing by forming parts using a specific type of sheet metal.

To define sheet metal parameters before you begin, access the **Sheet Metal Styles** tool by picking the **Sheet Metal Styles** button on the **Sheet Metal Features** panel bar, or picking the **Sheet Metal Styles** button on the **Sheet Metal Features** toolbar. This opens the **Sheet Metal Styles** dialog box in Figure 11-2, with these options available:

- **Style List.** This area lists all available sheet metal styles. Default is the only style unless you create additional sheet metal styles.
- **Active Style.** This drop-down list allows you to activate a sheet metal style. The sheet metal style that you want to use when creating sheet metal parts must be activated. To activate a sheet metal style, pick the drop-down list arrow and select the desired style.
- **Sheet.** This tab, shown in Figure 11-2, allows you to specify the sheet and flat pattern unfold parameters for the sheet metal part. The following options are located in the **Sheet** tab:

Figure 11-2.
The **Sheet Metal Styles** dialog box, **Sheet** tab.



- **Sheet.** This area allows you to select the type of material you want to use for the sheet metal part by selecting the **Material** drop-down list and choosing the desired material. You can also define the material thickness by entering or selecting a value using the **Thickness** edit box.

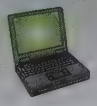
NOTE



The type of material can also be defined in the **Physical** tab of the **Properties** dialog box.

- **Flat Pattern.** This area allows you to define the flat pattern unfold options for the sheet metal part. Select either the **Linear** or **Bend Table** unfold method option from the **Unfold Method** drop-down list. When you select the **Linear** option, you can specify an unfold method value from the **Unfold Method Value** drop-down list. When you select the **Bend Table** option, the **Open** dialog box appears, allowing you to search for an existing bend table, which is a text file.

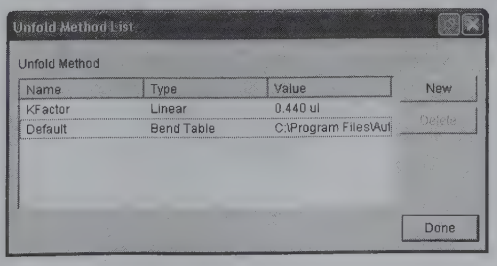
PROFESSIONAL TIP



To create your own bend tables or customize a bend table, use a spreadsheet. Then save the spreadsheet as a .txt file for use in the sheet metal environment.

If you want to change, add, or remove an unfold method value, pick the **Modify List** button to open the **Unfold Method List** dialog box. See Figure 11-3. To change the name of an unfold method value, pick the current name and enter a new value. You can change

Figure 11-3.
The **Unfold Method List** dialog box.



the type of unfold method by selecting the current type, and picking either **Linear** or **Bend Table** from the **Type** drop-down list. To modify the value, pick the current value and enter the desired value between 0 and 1, or select a value from the list. To create a new unfold method value, pick the **New** button and create a new unfold method value with the desired name, type, and value using the options previously described. You can also remove unfold methods by selecting the name you want to remove, and picking the **Delete** button.

When you are finished specifying the sheet parameters, select another tab to continue defining the sheet metal style, or pick the **Save** button to save changes to the style, and select the **Done** button to exit the **Sheet Metal Styles** dialog box.

- **Bend.** This tab allows you to specify the sheet metal part, bend parameters, and has the following areas shown in Figure 11-4:
  - **Radius.** This edit box allows you to specify the bend radius. The default bend radius is equal to the thickness of the material. You can override the bend radius setting for certain features when they are created, by entering or selecting a new radius.
  - **Relief Shape.** This drop-down list allows you to select the required bend relief shape. *Bend relief* is typically added to a sheet metal part to relieve stress, or the tear, that occurs when a portion of a piece of material is bent. You can pick either a straight, round, or no bend relief shape by selecting the **Straight**, **Round**, or **None** option. See Figure 11-5.
  - **Minimum Remnant.** This edit box allows you to specify the amount of sheet metal material between the inside relief edge and the feature edge. The default minimum remnant is twice the material thickness. If the size of the remnant is less than the value specified, the remnant is removed. See Figure 11-6.
  - **Relief Width.** This edit box allows you to define the width of the bend relief. The default relief width is the same as the material thickness. However, you can define a different relief width, with a value greater than 0. See Figure 11-7.
  - **Relief Depth.** This edit box allows you to specify the depth of the bend relief. The default relief depth is half the material thickness. However, you can specify a different relief depth, with a value greater than or equal to 0. See Figure 11-8.

Figure 11-4.
The **Bend** tab of the **Sheet Metal Styles** dialog box.

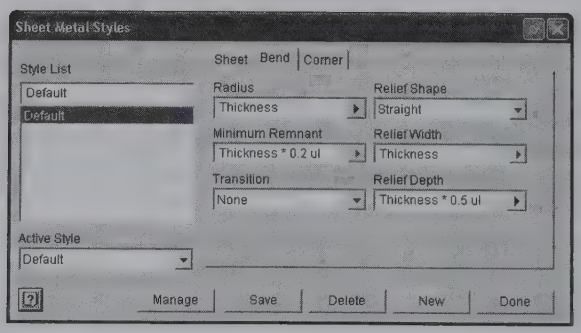


Figure 11-5.
An example of a straight bend relief, a round bend relief, and no bend relief.

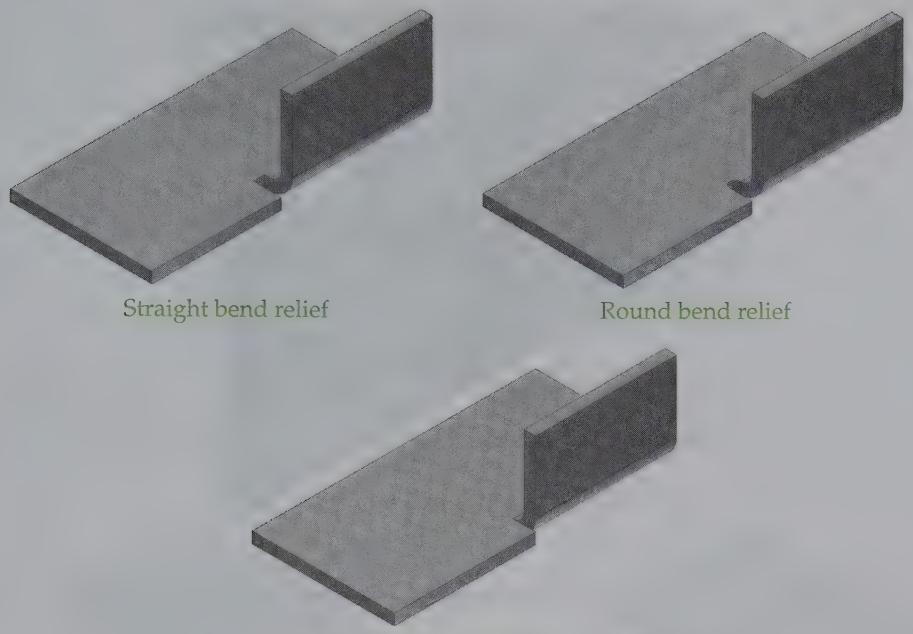


Figure 11-6.
An example of a remnant and a removed remnant.

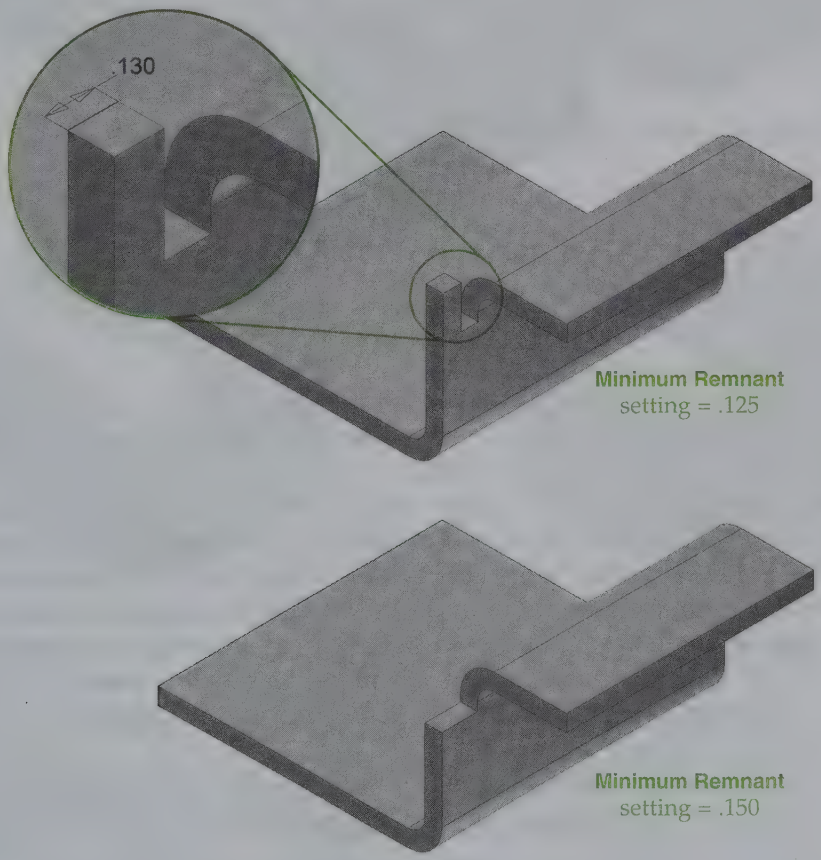


Figure 11-7.
An example of a relief width greater than the material thickness.

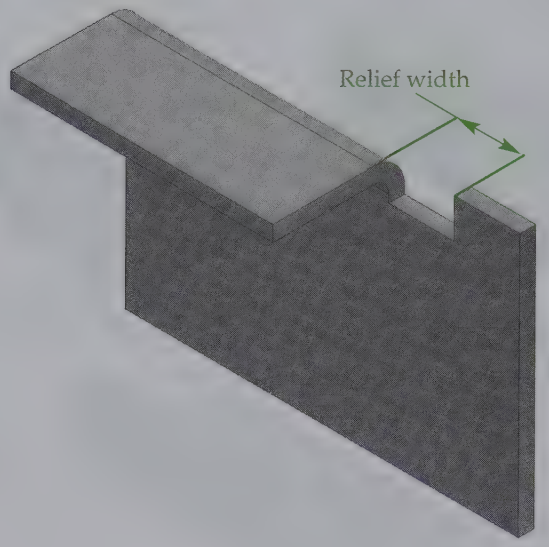
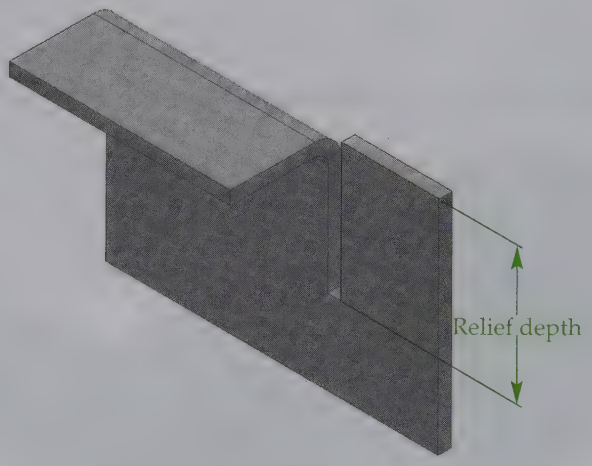


Figure 11-8.
An example of a relief depth greater than 0.



- **Transition.** This drop-down list allows you to select one of five different types of bend transitions to use when the sheet metal part is unfolded and when no bend relief is specified.

- **Corner.** This tab, shown in Figure 11-9, allows you to specify the sheet metal part corner relief parameters. *Corner relief* is typically added to a sheet metal part to relieve stress at a bend corner where three or more intersecting faces are created during a bend operation. The following are located in the **Corner** tab:
  - **Relief Shape.** This drop-down list allows you to specify the corner relief shape. You can specify a round, square, tear, trim to bend, or linear weld corner relief. The **Trim to Bend** relief shape option specifies no corner relief. See Figure 11-10.
  - **Relief Size.** This edit box allows you to specify the size of the corner relief. The default relief size is four times larger than the specified thickness of the material.

To develop a new sheet metal style, pick the **New** button. A copy of the active sheet metal style is displayed in the **Style List** edit box. Change the name of the style by picking

Figure 11-9.
The **Corner** tab of the **Sheet Metal Styles** dialog box.

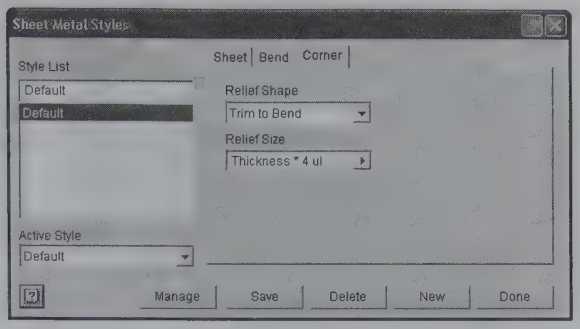
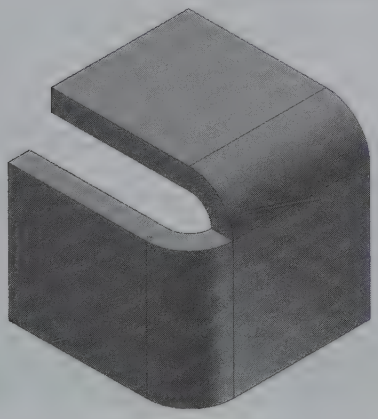
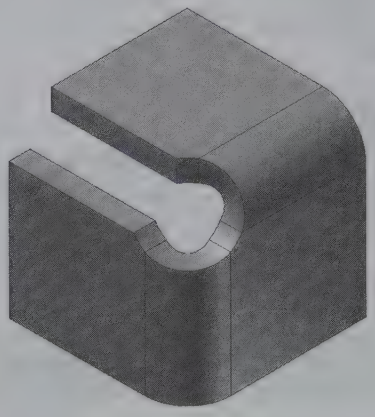


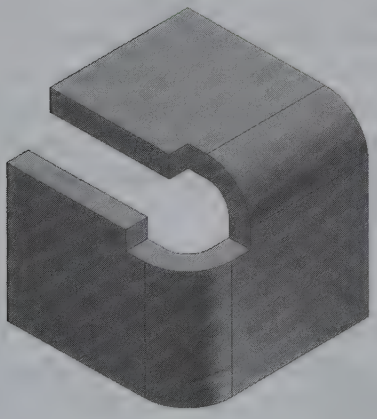
Figure 11-10.
These corner relief shapes are used to relieve stress and make bending a part easier.



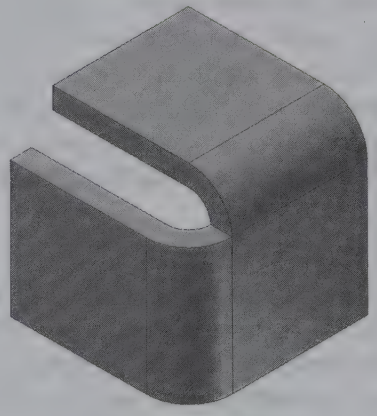
No corner relief



Round corner relief



Square corner relief



Torn corner relief

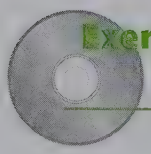
the default style name and entering the desired name. Then specify all the sheet metal options, using the previously described techniques. Once you fully define the sheet metal style, pick the **Save** button to save the style. Continue adding new styles as needed. Pick the **Manage** button to access the **Manage Sheet Metal Styles** dialog box, which can be used to import sheet metal files from an existing file into the current file. If you want to remove a style from the list, pick the style from the **Style List** box and select the **Delete** button.

When you finish modifying, adding, or removing sheet metal styles, pick the **Done** button to exit the **Sheet Metal Styles** dialog box.

NOTE



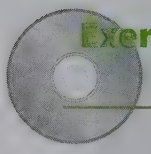
Even after you complete a sheet metal part, you can modify the sheet metal style options, located in the **Sheet Metal Styles** dialog box, and update the part to reflect the changes. Changes to sheet metal parameters can also be made in the **Parameters** dialog box.



Complete the Exercise on the Student CD.



Complete the Exercise on the Student CD.



Complete the Exercise on the Student CD.

Sheet Metal Sketched Features

Once you create a sketch, you are ready to develop a three-dimensional sheet metal part using Inventor sheet metal sketched feature tools. As in most other part applications, usually the first type of sheet metal feature you create is a sketched feature. Typically a face, created using the **Face** tool, is used as the initial base feature. Additional features then add material to, or take material from, the face. Sketches and sheet metal sketched features can be created anytime during sheet metal part model development. Sheet metal sketched features include faces, contour flanges, cuts, and folds.

Creating Faces

A ***sheet metal face*** is similar to an extrusion, and is typically used to create a base sheet metal feature. An example of a sheet metal face is shown in Figure 11-11. Once you develop sketch geometry to use for the sheet metal face, access the **Face** tool by picking the **Face** button on the **Sheet Metal Features** panel bar or picking the **Face** button on the **Sheet Metal Features** toolbar. This opens the **Face** dialog box, shown in Figure 11-12. When you create a base feature face, typically you need to specify only the sketch profile and offset. However, if you are creating additional faces, more options are available. The tools required to generate an initial base feature face are located in the **Shape** area of the **Shape** tab. This tab contains the face shape definition options. If only one sketch loop exists, the sketch profile is already selected. To create a face from a sketch with more than one profile available, pick the **Profile** button from the **Shape** area of the **Shape** tab, if it is not already active. Then pick the desired face profile sketch. When the sketch profile is selected, a preview of the face is displayed. See Figure 11-13. The next step is to determine which side of the sketch plane you

Figure 11-11.
An example of a simple sheet metal face, created from a sketched rectangle.

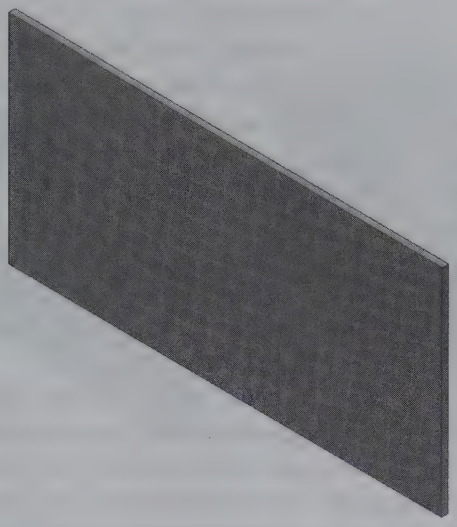


Figure 11-12.
The **Face** dialog box, **Shape** tab.

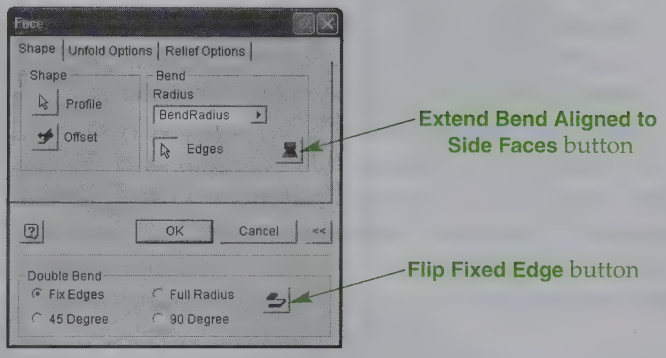
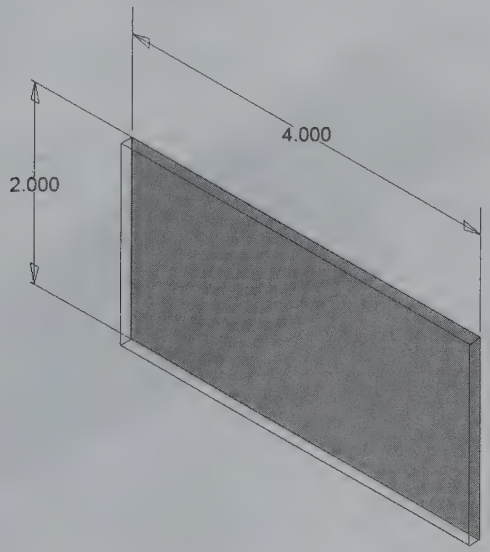


Figure 11-13.
A sheet metal face preview.



want the face depth to occur. If you want the face to be created on the opposite side of the current preview, pick the **Offset** button. If you are creating only one face, no other faces are present, and you do not need to define any other sheet metal face options, pick the **OK** button to generate the sheet metal face.

More options are available in the **Face** dialog box when you add additional faces to the base feature face. The additional face shape options are found in the **Bend** area of the **Shape** tab, the **Unfold Options** tab, and the **Relief Options** tab. To add a face to an existing face, select the sketch profile as shown in Figure 11-14 and choose the desired offset. Then define the face bend using the following options found in the **Bend** area of the **Shape** tab:

- **Radius.** This edit box allows you to define the bend radius. The default bend radius specified in the **Sheet Metal Styles** dialog box is described as "BendRadius" in the **Radius** edit box. However, you can override the default value by entering an appropriate bend radius.
- **Edges.** This button is used to select edges when defining a double bend. A double bend allows you to generate a bend between two parallel, noncoplanar faces. See Figure 11-15. To create a double bend, pick the **Edges** button and select the edge from which you want to bend. Once you select this edge, pick the **More** button to access the **Double Bend** area of the **Face** dialog box. Select the type of double bend you want to create. See Figure 11-16 for the various types of double bends. If the double edge preview does not look correct, you may need to modify the selected edges using the **Flip Fixed Edge** button. See Figure 11-17.
- **Extend Bend Aligned to Side Faces.** Pick this button to add material to a double bend face. When you select the **Extend Bend Aligned to Side Faces** button, the initial sketched face location is extended to the intersection of the selected edge. See Figure 11-18.

The next step in creating a sheet metal face is to modify the unfold and bend relief options located in the **Unfold Options** and **Relief Options** tabs of the **Face** dialog box, if necessary. The options available inside these two tabs are the same as those located in the **Sheet Metal Styles** dialog box. The preferences in **Unfold Options** and **Relief Options** tabs allow you to override the default options specified in the **Sheet Metal**

Figure 11-14.
Adding a sheet metal face to an existing face.

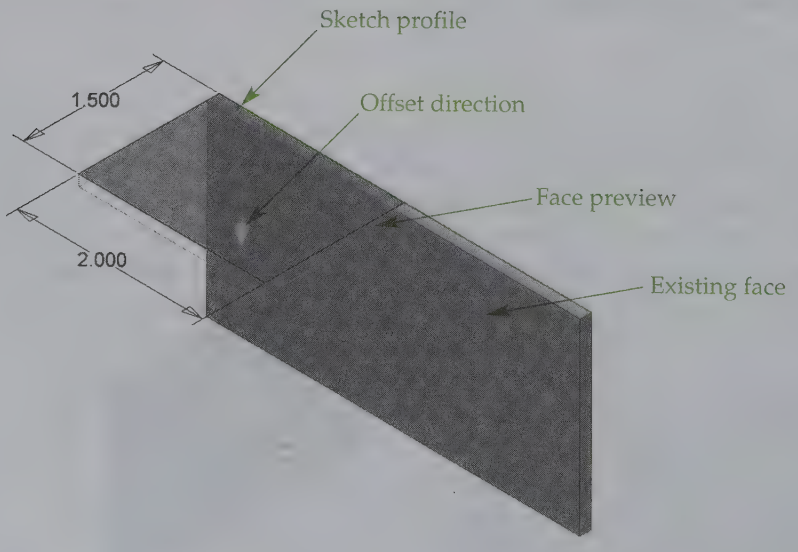


Figure 11-15.
Two parallel, noncoplanar sheet metal faces. Creating a double bend face, **Fix Edges** option.

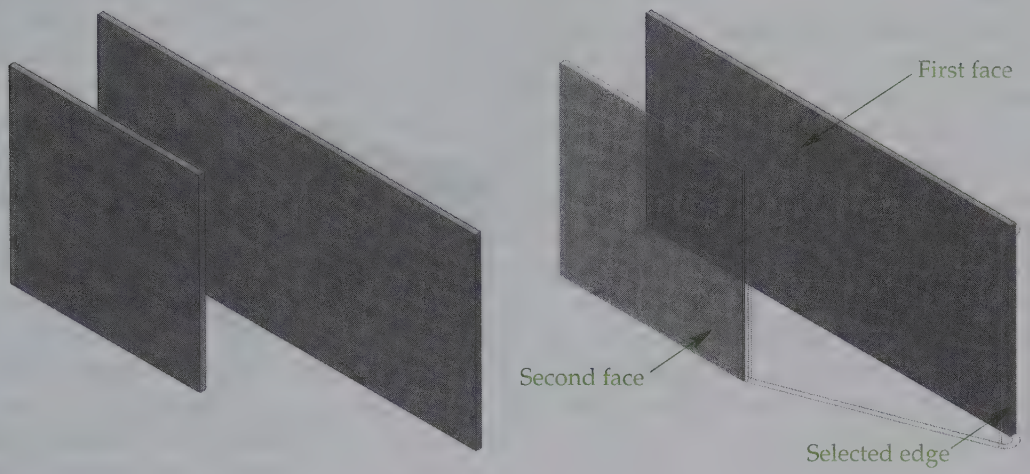
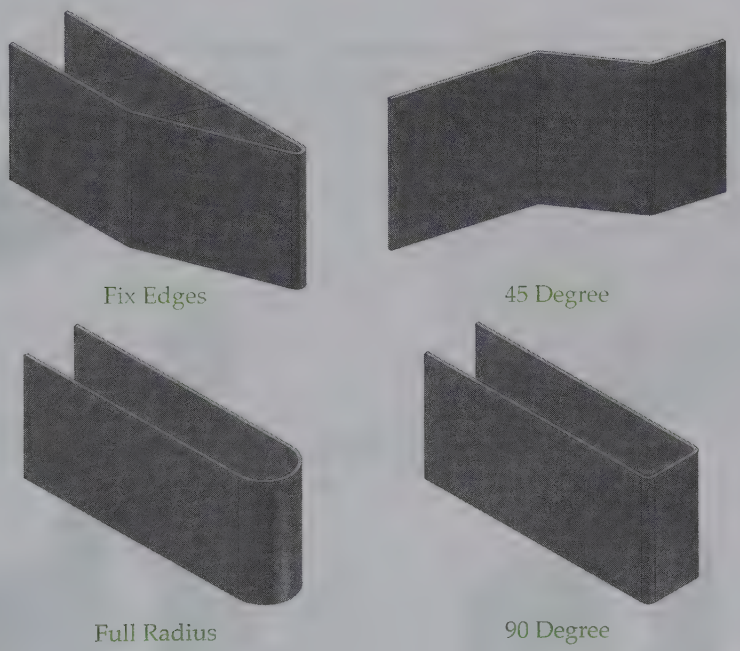


Figure 11-16.
These are the variety of double bend types available in Inventor.



Styles dialog box. However, if you want to use the sheet metal settings defined in the **Sheet Metal Styles** dialog box, select the **Default** option from the specified style parameter drop-down list. When you specify all the sheet metal face characteristics, pick the **OK** button to complete the face.



Exercise 11-4 Complete the Exercise on the Student CD.

Figure 11-17.
This is an example of flipping the fixed edges when creating a double bend.

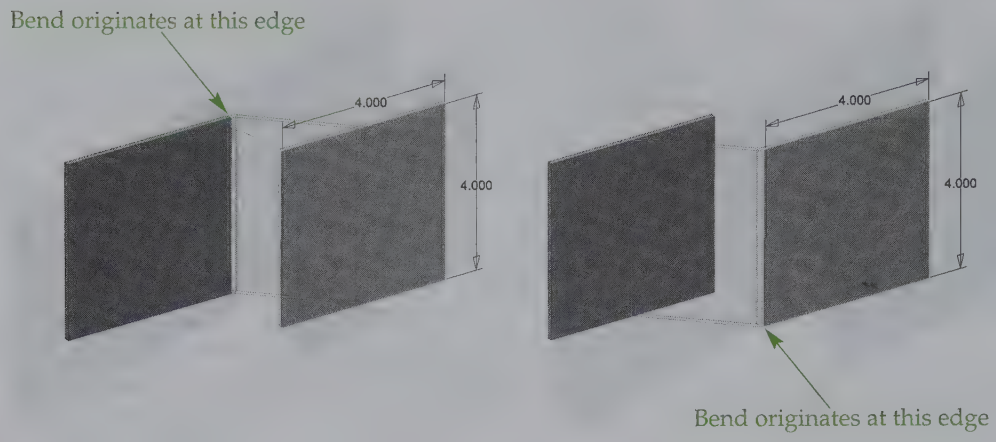
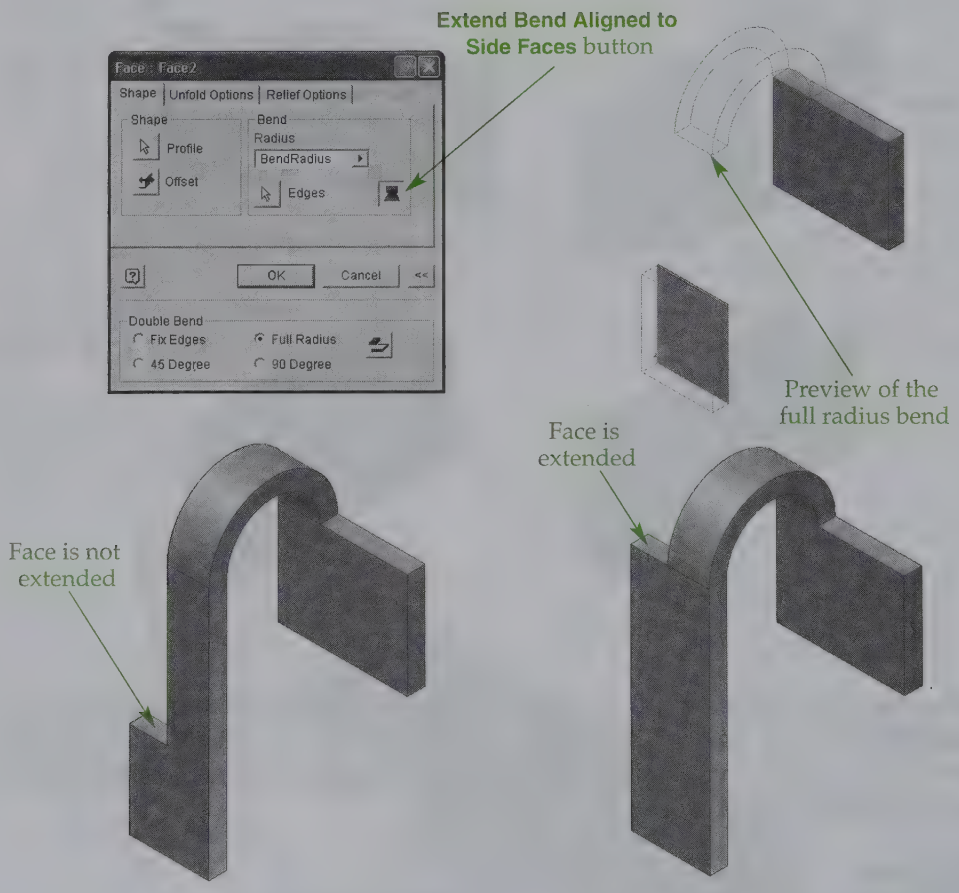


Figure 11-18.
When the **Extend Bend Aligned to Side Faces** button is selected, the face will move to meet the bend.



Creating Contour Flanges

A sheet metal ***contour flange*** is similar to a face, but is typically created using an open sketch profile, such as the sketch shown in Figure 11-19. By default, a sheet metal contour flange is created using the parameters specified in the **Sheet Metal Styles** dialog box. However, when you create individual contour flanges, you can override

the bend, unfold, and bend relief options. All other options must be defined in the **Sheet Metal Styles** dialog box or updated using the **Parameters** dialog box. Contour flanges can be used quickly to form a variety of sheet metal parts, and are often used to replicate extruded sheet metal features, such as gutters, corrugated panels, or the portion of a frame shown in Figure 11-20. Once you develop sketch geometry to use for the sheet metal contour flange, access the **Contour Flange** tool by picking the **Contour Flange** button on the **Sheet Metal Features** panel bar, or picking the **Contour Flange** button on the **Sheet Metal Features** toolbar. This opens the **Contour Flange** dialog box in Figure 11-21. When you create a base feature contour flange, typically you need to specify only the sketch profile and length of the feature using the **Distance** extents option. However, if you are creating additional contour flanges or other sheet metal features are present, more options are available.

To create a contour flange, first specify the shape characteristics in the **Shape** tab of the **Contour Flange** dialog box. Initially, the **Profile** button should be selected, but if not, pick the **Profile** button and choose the sketch profile used to create the contour flange. When the sketch profile is selected, a preview of the contour flange is displayed. See Figure 11-22.

Next, if necessary, define the contour flange offset by selecting the **Offset** button. The offset determines whether the depth of the sheet metal thickness is applied to the inside or the outside of the sketch profile. Once you define the contour flange offset,

Figure 11-19.
An example of an open sketch and the contour flange generated by it.

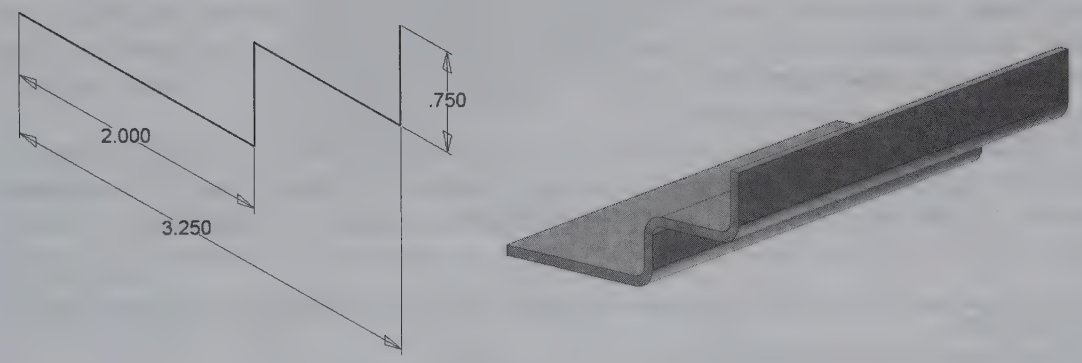


Figure 11-20.
An example of a sheet metal contour flange, created from an open sketch profile.

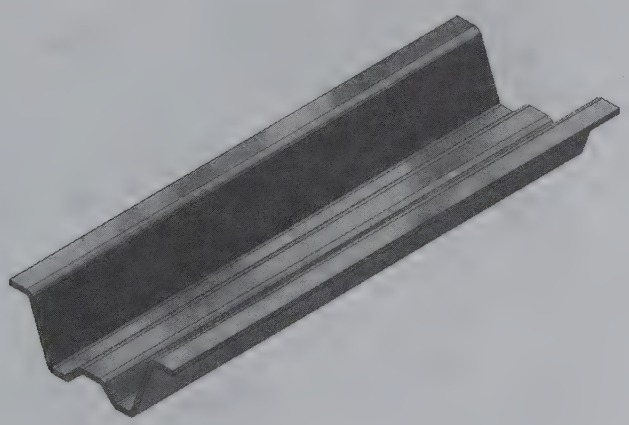
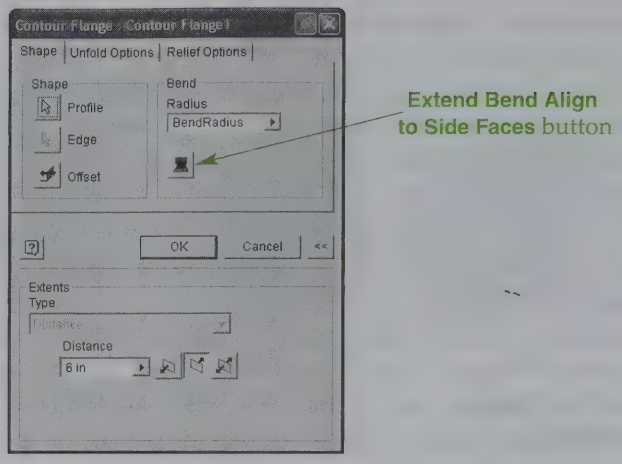


Figure 11-21.
The **Contour Flange** dialog box, **Shape** tab.

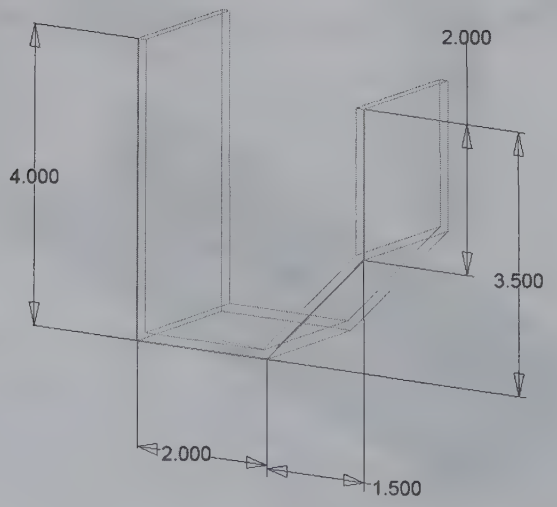


specify the flange bend radius by entering or selecting a value in the **Radius** edit box. The default bend radius, specified in the **Sheet Metal Styles** dialog box, is identified as BendRadius in the **Radius** edit box. However, you can override the default value by entering an appropriate bend radius.

The next step in creating a sheet metal contour flange is to modify the unfold and bend relief options located in the **Unfold Options** and **Relief Options** tabs of the **Contour Flange** dialog box, if necessary. The options available are the same as those located in the **Sheet Metal Styles** dialog box. The preferences in the **Unfold Options** and **Relief Options** tabs allow you to override the default options specified in the **Sheet Metal Styles** dialog box. However, if you want to use the sheet metal settings defined in the **Sheet Metal Styles** dialog box, select the **Default** option from the specified style parameter drop-down list.

When you create a base feature and no other sheet metal features are present, the only way to define the length, or depth, of the contour flange is with the **Distance** extents option. However, you can use the **Distance** extents option anytime to specify

Figure 11-22.
A sheet metal contour flange preview.



the length of the flange. To define the extents of the flange using the **Distance** extents option, access the **Extents** area by selecting the **More** button, if it is not already selected. Then, choose the **Distance** option from the **Type** drop-down list. Next, enter the flange length, or choose a length in the **Distance** edit box. Finally, specify on which side of the sketch you want the flange to occur by selecting the appropriate direction button. An example of a distance extents contour flange is shown in Figure 11-23.

In addition to the **Distance** extents option, the following extents specifications are available when other sheet metal contour features are present, such as adding a contour flange to a face:

- **Edge.** This extents option allows you to specify an existing sheet metal feature edge as the length of the contour flange. To use the **Edge** extents option, first select the sketch profile. Then pick the **Edge** button, if it is not already active, followed by selecting the desired existing feature edge. As shown in Figure 11-24, selectable edges are perpendicular to the sketch plane.

Figure 11-23.
A 12″ contour flange created using the **Distance** extents option.

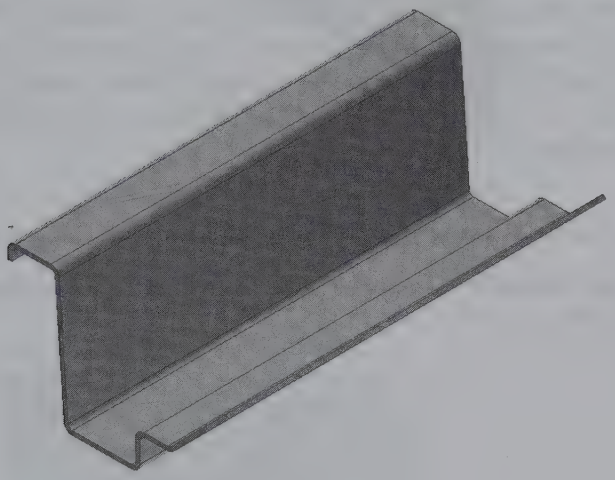
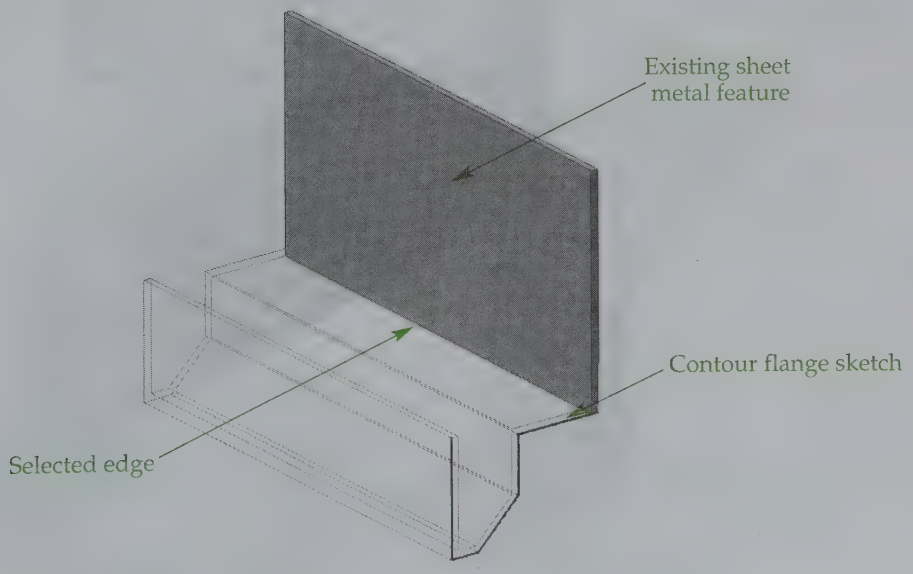


Figure 11-24.
Using the **Edge** extents option.



- **Width.** This extents option allows you to specify the width of the contour flange and an offset distance from a selected point. See Figure 11-25. To use the **Width** extents option, select the sketch profile, pick the **Edge** button—if it is not already active, and select the desired existing feature edge. Then pick the **More** button to access the **Extents** area of the **Contour Flange** dialog box, and select the **Width** option from the **Type** drop-down list. Next, pick the **Select Start Point** button—if it is not active, and pick an endpoint on the existing feature. The selected point identifies from where the contour flange is offset. Enter or select an offset value using the **Offset** edit box. If the offset occurs on the wrong side, pick the **Flip Direction** button to flip the direction. Finally, specify the width of the contour flange by entering or selecting a value using the **Width** edit box.
- **Offset.** This extents option allows you to specify the width of the contour flange, by defining the offset distance from two selected points. To use the **Offset** extents option, select the sketch profile, pick the **Edge** button—if it is not already active, and select the desired existing feature edge. Then pick the **More** button to access the **Extents** area of the **Contour Flange** dialog box, and select the **Offset** option from the **Type** drop-down list. See Figure 11-26. Next, pick the **Select Start Point** button, if it is not active, and pick a point on the existing feature where you want the contour flange offset to begin. Once the start point is selected, the **Select End Point** button should become active. If not, pick the **Select End Point** button and choose a point on the existing feature where you want the contour flange offset to end. Finally, enter or select a value in the **Offset1** and **Offset2** edit boxes.

Figure 11-25.
The **Width** option of the **Extents** area requires a distance to offset the contour flange and a measurement for the flange width.

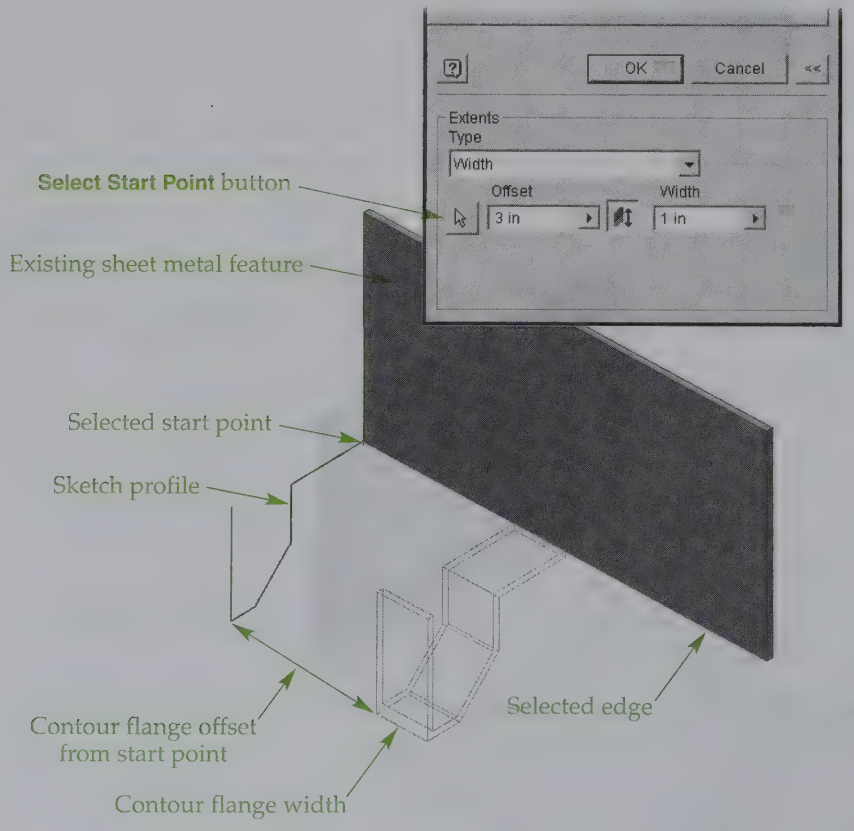
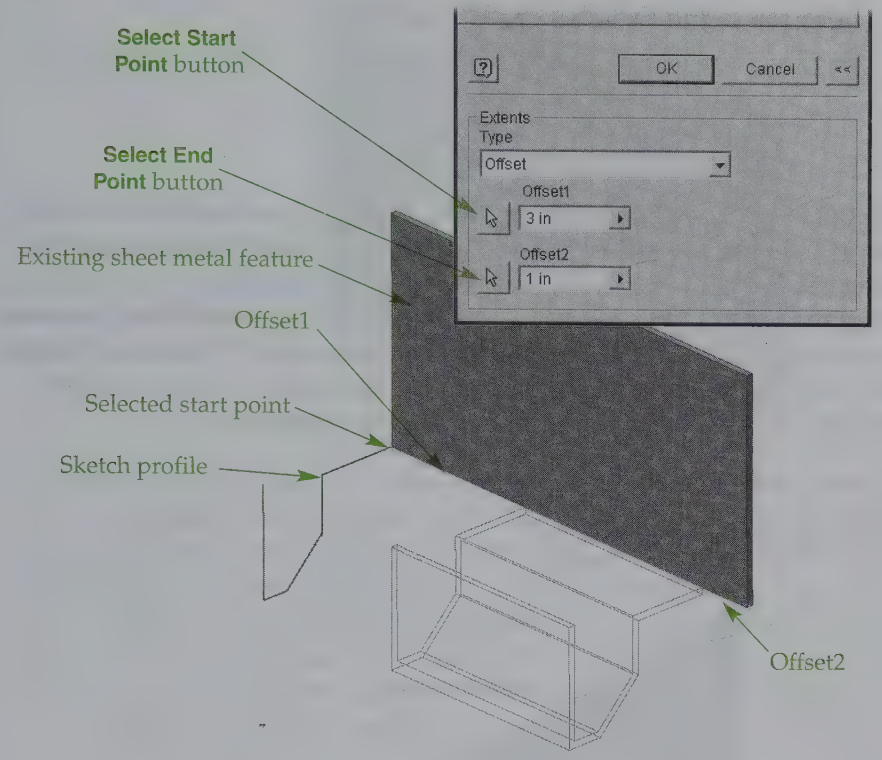
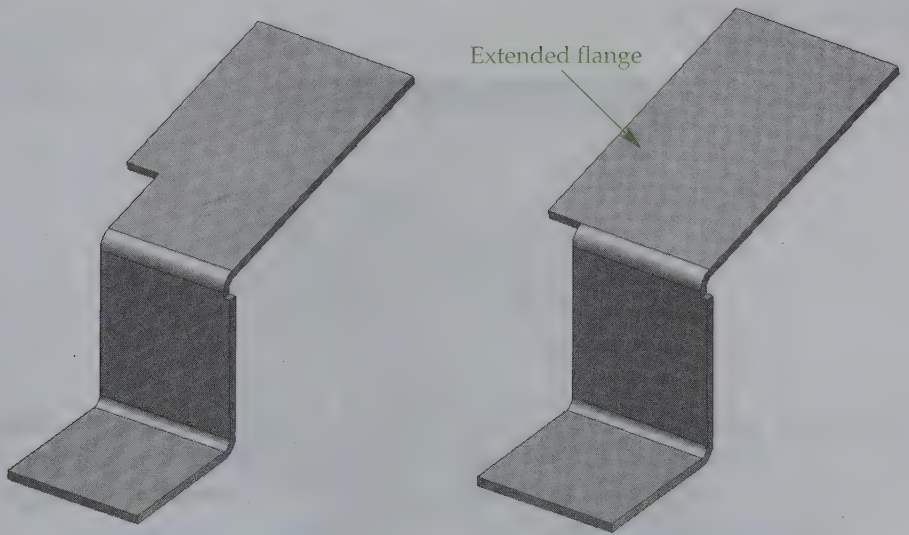


Figure 11-26.
Using the **Offset** extents option allows the flange width to change while maintaining a specified distance between the flange sides and selected geometry.



When adding contour flanges to existing sheet metal features, you may need to select the **Extend Bend Aligned to Side Faces** button, located in the **Bend** area of the **Contour Flange** dialog box. Pick this button to add material to the new contour flange. When you select the **Extend Bend Aligned to Side Faces** button, the initial sketched flange location is extended to the intersection of the selected edge. See Figure 11-27.

Figure 11-27.
An example of a contour flange created without selecting the **Extend Bend Aligned to Side Faces** button and with the **Extend Bend Aligned to Side Faces** button selected.



Extend Bend Aligned to Side Faces is off

Extend Bend Aligned to Side Faces is on

When you specify all the sheet metal contour flange options, pick the **OK** button to complete the contour flange.



Complete the Exercise on the Student CD.

Developing Cuts

The sheet metal **Cut** tool is used to remove material through a selected sketch profile on an exiting sheet metal feature. Using the **Cut** tool is similar to using the **Extrusion** tool with the cut operation. See Figure 11-28. Once you develop sketch geometry to use for the sheet metal cut, access the **Cut** tool by picking the **Cut** button on the **Sheet Metal Features** panel bar, or picking the **Cut** button on the **Sheet Metal Features** toolbar. This opens the **Cut** dialog box in Figure 11-29. To use the **Cut** tool, pick the sketch profile you want to use for the cut. Then specify the cut depth using one of the following techniques:

Figure 11-28.
An example of a cut through a sheet metal face.

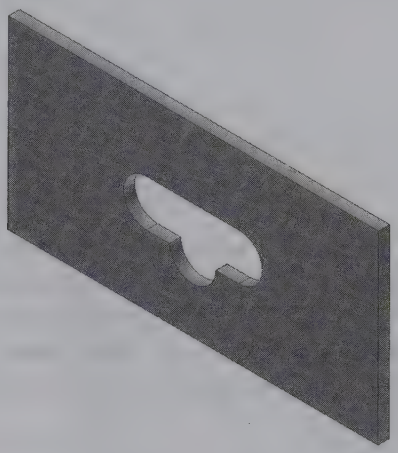
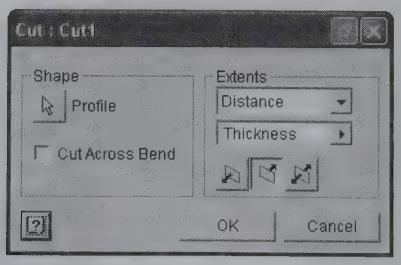
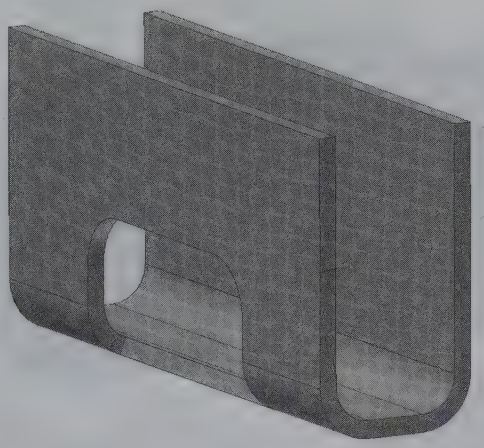


Figure 11-29.
The **Cut** dialog box, **Distance** extents option.



- **Cut Across Bend.** This check box, located in the **Shape** area of the **Cut** dialog box, allows you to cut through the thickness of the sheet metal and the full length of the bend. See Figure 11-30. Other extent operations are not available when the **Cut Across Bend** check box is selected.

Figure 11-30.
An example of a cut created using the **Cut Across Bend** option.



- **Distance.** This option, located in the **Extents** area of the **Cut** dialog box, allows you to define the amount of material the cut removes, using a specified distance. See Figure 11-31. To use the **Distance** extent option, pick **Distance** from the **Extent Type** drop-down list, and specify the distance in the **Distance** edit box. The default distance is "Thickness," which is the thickness of the material specified in the **Sheet Metal Styles** dialog box. To modify the distance, enter a new-value or select a value in the **Distance** edit box.
- **To Next.** Selecting this option, located in the **Extents** area of the **Cut** dialog box, enables you to cut a sketch profile to the next possible face or plane. See Figure 11-32. To cut a sketch profile to the next face or plane, pick the sketch profile and select the direction you want the cut to occur.

NOTE



There is no equal side, or midplane, direction option available when cutting a **To Next** face or plane. If a face or plane does not exist that can end the extrusion, the feature cannot be created.

Figure 11-31.
An example of a cut created using the **Distance** extent option five times the material thickness.

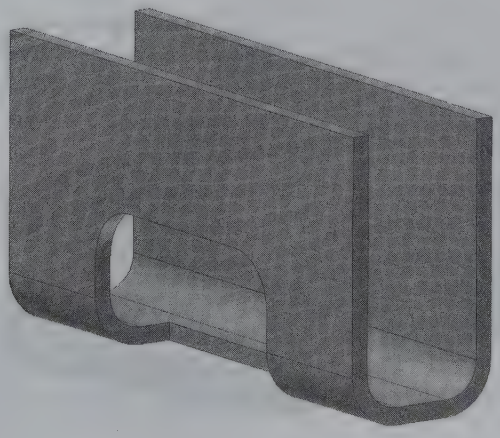
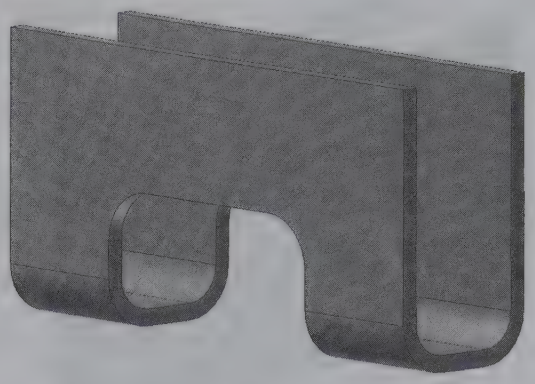


Figure 11-32.
Cutting through sheet metal using the **To Next** extents option.



- **To.** This option is similar to the **To Next** option previously described. However, selecting **To** allows you to choose a face or plane to end the cut, instead of the cut ending at the next possible face or plane. See Figure 11-33. To terminate a cut at a specified face or plane using the **To** option, pick the face or plane to which you want the cut to terminate. You may need to choose the **Select surface to end the feature creation** button, if it is not already activated. In addition, if the face or plane you choose does not intersect the extrusion path, you need to pick the **Terminate feature on extended face** check box.
- **From To.** This option is similar to the **To** option previously described. However, selecting **From To** allows you to choose a face or plane to begin and end the cut, instead of just the ending surface. See Figure 11-34. To begin and terminate a cut at a specified surface using the **From To** option, pick the face or plane from which you want to begin the cut. You may need to choose the **Select surface to start the feature creation** button, if it is not already activated. Next, pick the face or plane where you want to end the cut. Again, you may need to choose the **Select surface to end the feature creation** button, if it is not already activated. If the surfaces you choose to start and end the cut do not intersect the cut path, you need to check the **Terminate feature on extended face** check box.

Figure 11-33.
The **To** extents option allows the cut to continue to a specified surface or an extension of a surface.

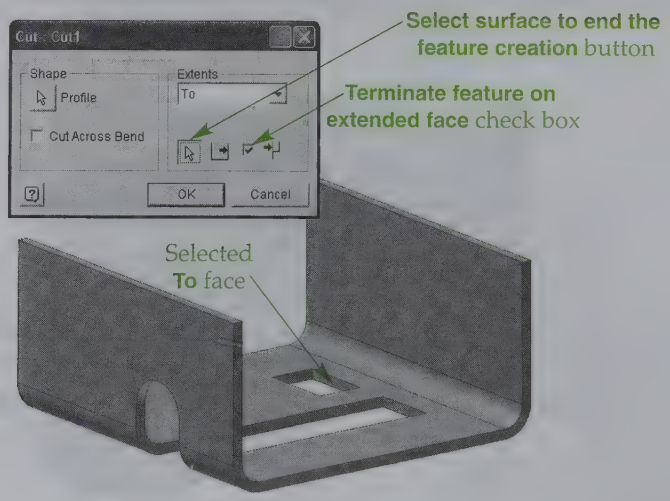
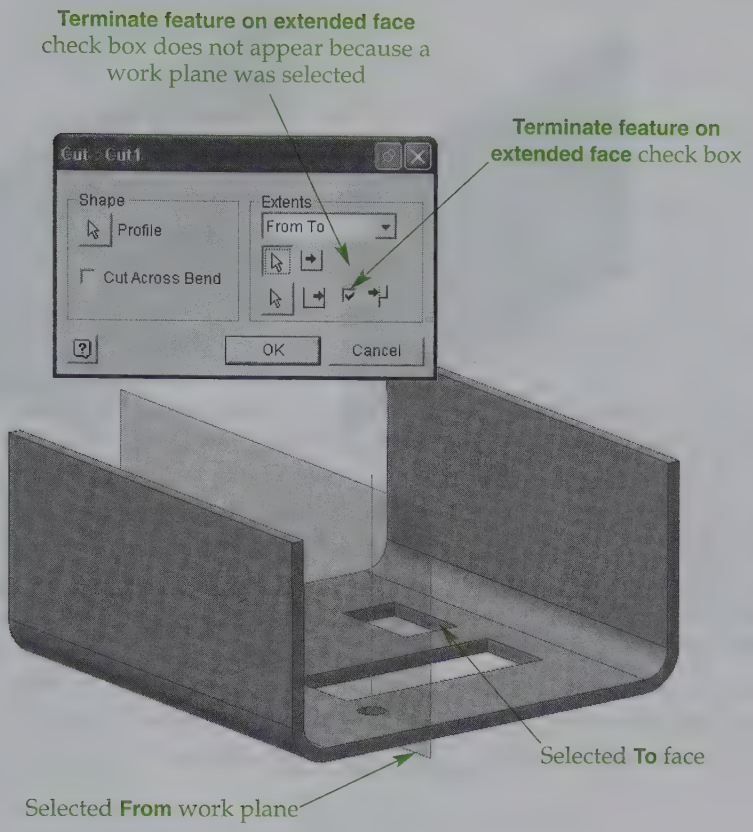


Figure 11-34.
The **From To** extents option allows you to control where the cut starts and ends.



- **All.** Selecting this option allows you to cut a sketch profile directly through all other features. If you know for sure you want to cut the entire way through existing features, use the **All** option. It does not require any distances or surface selections, and can save some time. To cut a feature through all existing features, pick the sketch profile and select the direction you want the cut to occur.

When you specify all the sheet metal cut options, pick the **OK** button to complete the cut.

Exercise 11-6 Complete the Exercise on the Student CD.

Creating Folds

Sheet metal *folds* are created by folding a section of another sheet metal feature, such as a face or contour flange, along a sketched line. See Figure 11-35. The first step in creating a fold is to make a sketch. Most of the time, a single sketched line is used to define the fold edge. Once you develop a sketch, access the **Fold** tool by picking the **Fold** button on the **Sheet Metal Features** panel bar or picking the **Fold** button on the **Sheet Metal Features** toolbar. The **Fold** dialog box in Figure 11-36 is initially opened in the **Shape** tab with the following options:

- **Bend Line.** This button allows you to pick the sketched line where you want the fold to occur.
- **Bend Radius.** This edit box is used to define the bend radius. The default bend radius specified in the **Sheet Metal Styles** dialog box is identified as

Figure 11-35.
An example of a simple sheet metal fold that uses a sketched line to control the fold's location.

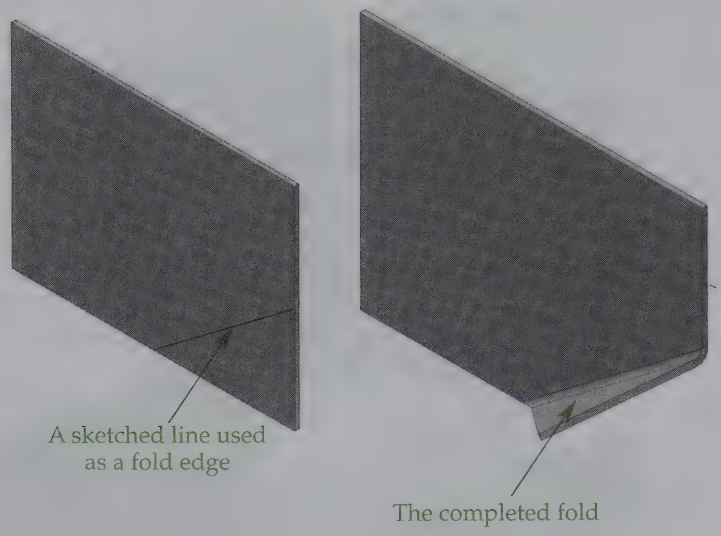
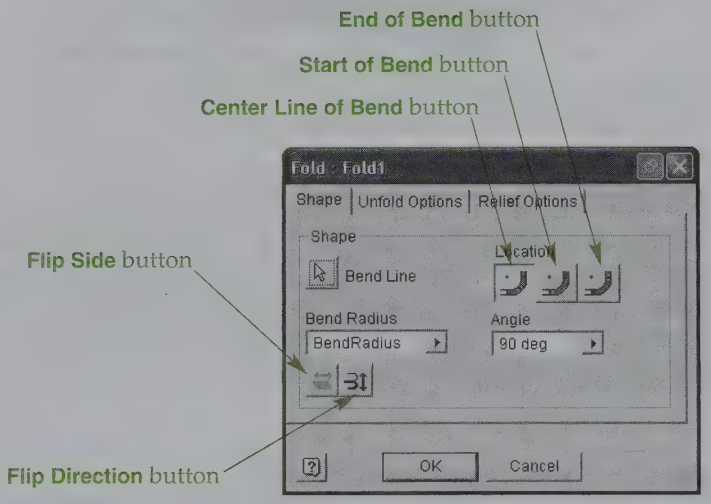


Figure 11-36.
The **Fold** dialog box, **Shape** tab.



"BendRadius" in the **Radius** edit box. However, you can override the default value by entering an appropriate bend radius.

- **Location.** The following three buttons control where the sheet metal material is folded in reference to the sketched line:
  - **Center Line of Bend.** This button uses the sketched line as a center line and bends the sheet metal material equally between the sketch lines.
  - **Start of Bend.** This button begins the bend at the sketched line and does not alter the material that is not bent.
  - **End of Bend.** This button ends the bend at the sketched line and does not alter the material that is bent.

NOTE

The displays on the three location buttons, and the preview of the operation on the part, help you determine the desired bend location.

- **Angle.** This edit box allows you to specify the amount of bend applied to the section of material being bent. The default bend is 90°, which results in a right-angle bend. However, you can enter any appropriate angle or select an angle from the list. See Figure 11-37.
- **Flip Side.** This button controls where the angle of the fold is applied. See Figure 11-38.
- **Flip Direction.** This button controls which side of the sheet metal feature the fold occurs. See Figure 11-38.

Once you define all the fold shape options, the next step in creating a sheet metal fold is to modify the unfold and bend relief specifications located in the **Unfold Options** and **Relief Options** tabs of the **Fold** dialog box, if necessary. The options available inside these tabs are the same as those located in the **Sheet Metal Styles** dialog box. The preferences in these tabs allow you to override the default options specified in the **Sheet Metal Styles** dialog box. However, if you want to use the sheet metal settings defined in the **Sheet Metal Styles** dialog box, select the **Default** option from the specified style parameter drop-down list.

When you specify all the sheet metal fold options, pick the **OK** button if you are generating only one fold. If you want to use the **Fold** tool again, pick the **Apply** button and continue producing folds as needed. When finished, pick the **OK** button to exit the tool.

Exercise 11-7 Complete the Exercise on the Student CD.

Figure 11-37.
The angles control the amount of bend in these folds.

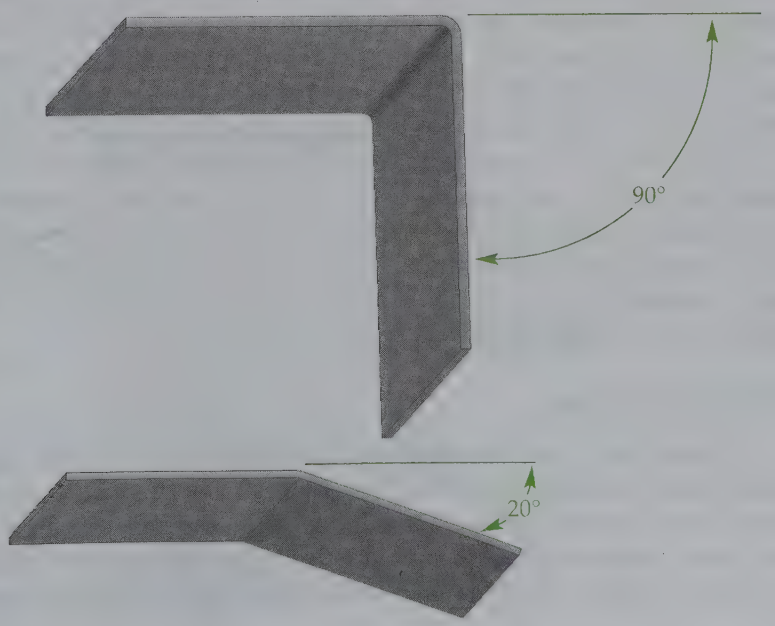
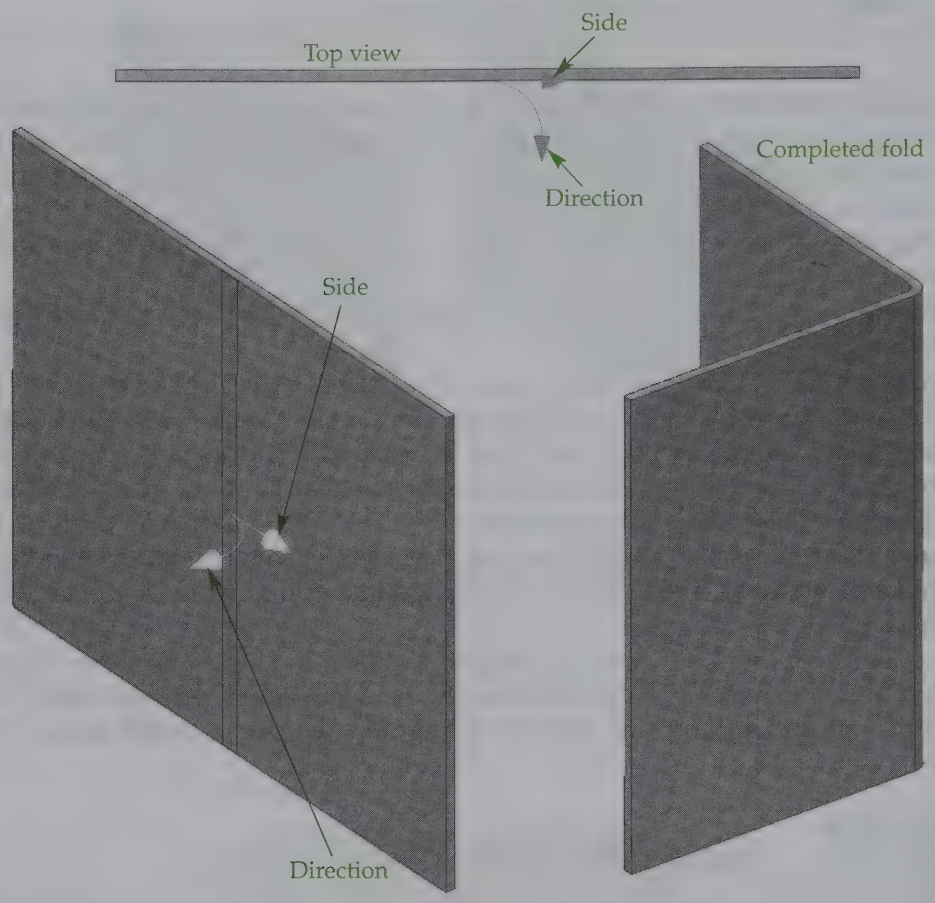


Figure 11-38.
Compare the preview of side and direction options, as seen in a top and isometric views, to the complete fold.



Sheet Metal Placed Features

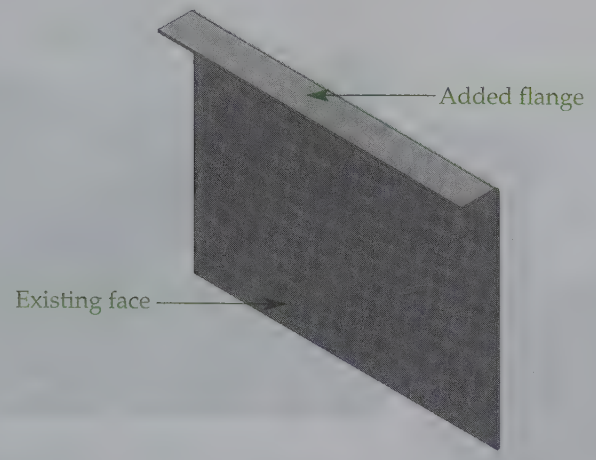
Once you generate at least one sheet metal sketched feature, you can further develop the part using Inventor sheet metal placed feature tools. ***Placed features*** can be added to the most basic features, such as a face or cut, or to very complex, multi-feature sheet metal parts. As the name implies, placed features are placed on existing features. For example, a corner round is placed on the corner of a contour flange.

In contrast to the sheet metal sketched features previously described, sheet metal placed features do not require a sketch to be created. However, they do require an acceptable sketched, placed, catalog, or pattern feature to be present. Placed features can be created anytime during sheet metal part development, as long as one of these suitable features exists. Sheet metal placed features include flanges, hems, corner seams, bends, corner rounds, and corner chamfers.

Adding Flanges

A *flange* is a type of sheet metal face that can be added to an existing sheet metal feature edge. See **Figure 11-39**. In contrast to a sheet metal face, flanges do not require a sketch and are typically less geometrically complex. Flanges are added to existing sheet metal feature edges and create specified shape and bend preferences between the existing feature and the placed flange. Flanges are created using the **Flange** tool by

Figure 11-39.
This is an example of a simple sheet metal flange.



picking the **Flange** button on the **Sheet Metal Features** panel bar, or picking the **Flange** button on the **Sheet Metal Features** toolbar. The **Flange** dialog box is displayed. See Figure 11-40.

To place a flange, pick the **Select Edge** button, if it is not already active, and pick the existing feature edge on which the flange will be created. The edge you select identifies the beginning of the flange length and the location of the bend. See Figure 11-41. Once you select the flange edge, use the **Flip Offset** button to identify where you want the flange depth to occur. The flange offset defines the beginning of the flange depth and the location of the bend.

Next, specify the flange length by entering or selecting a value in the **Distance** edit box. The flange distance specifies the flange length from the selected edge on the existing feature. You may need to select the **Flip Direction** button to reverse the direction of the flange length. The flange direction defines the side of the existing feature on which the flange is created and will also influence the beginning of the flange width and the location of the bend.

Figure 11-40.
The **Flange** dialog box, **Shape** tab and **Edge** extents option.

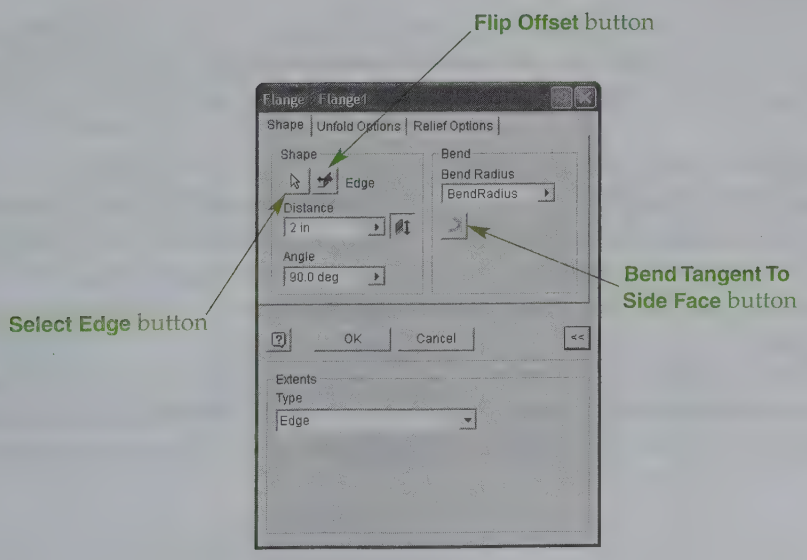
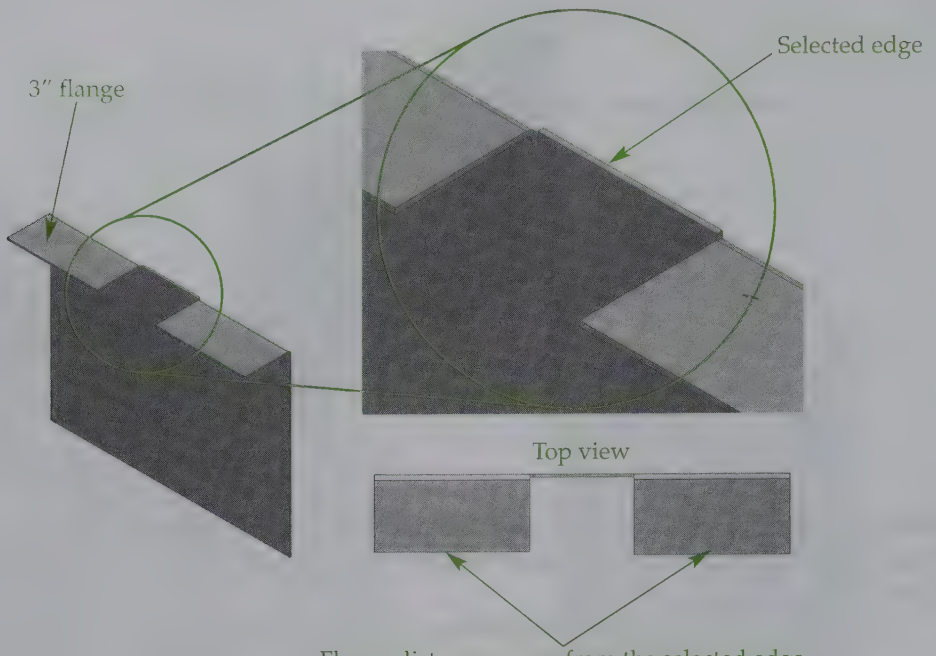


Figure 11-41.
The effects of choosing certain feature edges.



After you determine the flange distance, specify the angle of the flange by entering or selecting a value in the **Angle** edit box. Then specify the following options located in the **Bend** area of the **Flange** dialog box.

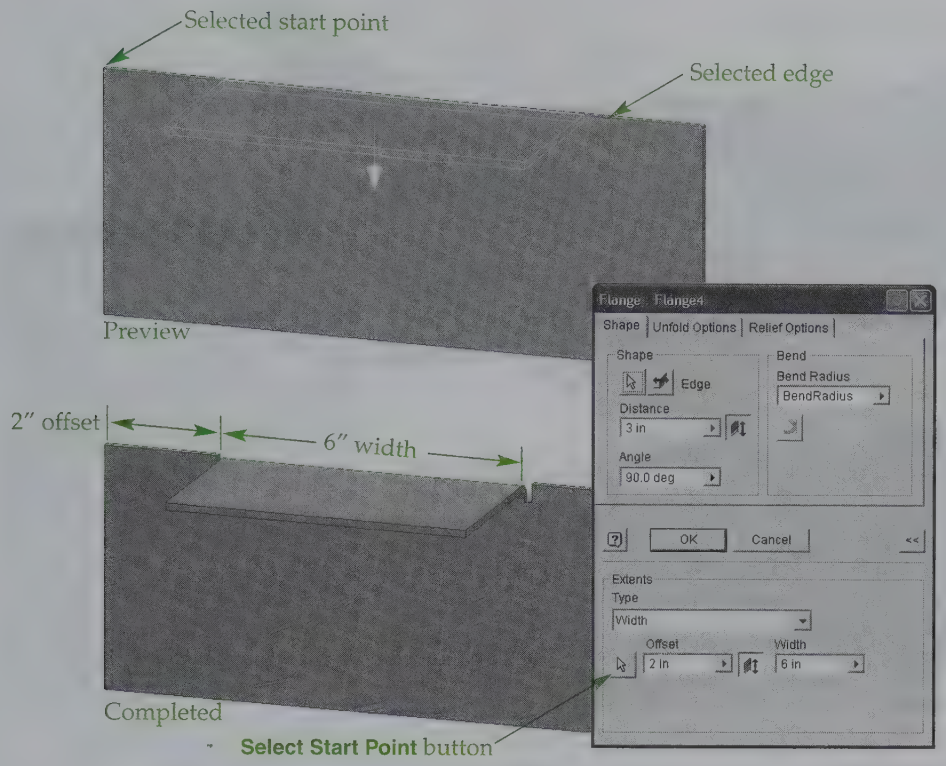
- **Bend Radius.** This edit box allows you to define the bend radius. The default bend radius specified in the **Sheet Metal Styles** dialog box is specified as BendRadius in the **Radius** edit box. However, you can override the default by entering any appropriate bend radius.
- **Bend Tangent To Side Face.** Pick this button to create a bend tangent to the selected flange edge.

Once you define all the flange shape options, you may want to modify the unfold and bend relief characteristics located in the **Unfold Options** and **Relief Options** tabs of the **Flange** dialog box, as previously described in the sheet metal sketched feature tools.

The final step in creating a sheet metal flange is to define the flange extents, using the **Extents** area of the **Flange** dialog box. This area is accessed by picking the **More** button. By default, the **Edge** extent type is active. This option, as shown back in Figure 11-39, creates a flange that extends the entire length of the selected feature edge. In addition to the **Edge** extents option, the following extents specifications are available when placing a sheet metal flange:

- **Width.** This extents option allows you to specify the width of the flange and an offset distance from a selected point. To use the **Width** extents option, select **Width** from the **Type** drop-down list. See Figure 11-42. Then, pick the **Select Start**

Figure 11-42.
When using the **Width** option, you must specify the offset distance and the width of the flange.



Point button, if it is not active, and pick a point on the existing feature edge. The selected point identifies from where the flange is offset. Next, enter or select an offset value using the **Offset** edit box. If the offset occurs on the wrong side, pick the **Flip Direction** button to flip the offset. Finally, specify the width of the flange by entering a value or selecting a value using the **Width** edit box.

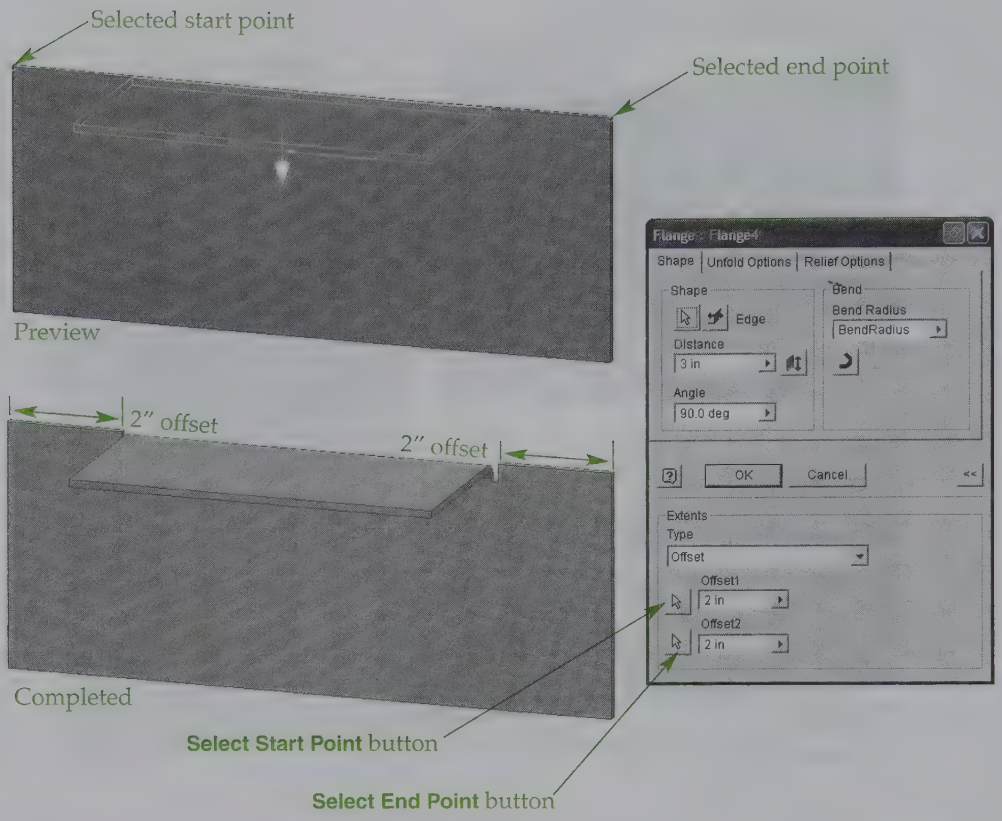
- **Offset.** This extents option allows you to specify the width of the flange by defining the offset distance from two selected points. To use the **Offset** extents option, select **Offset** from the **Type** drop-down list. See Figure 11-43. Next, pick the **Select Start Point** button, if it is not active, and pick a point on the existing feature where you want the flange offset to begin. Once the start point is selected, the **Select End Point** button should become active. If not, pick the **Select End Point** button and choose a point on the existing feature edge where you want the flange offset to end. Finally, enter or select a value from the **Offset1** and **Offset2** edit boxes.

When you specify all the sheet metal flange options, pick the **OK** button if you are generating only one flange. If you want to use the **Flange** tool again, pick the **Apply** button and continue producing flanges as needed. When finished, pick the **OK** button to exit the tool.



Complete the Exercise on the Student CD.

Figure 11-43.
When using the **Offset** option, you must specify the offset distances from each side of the flange.



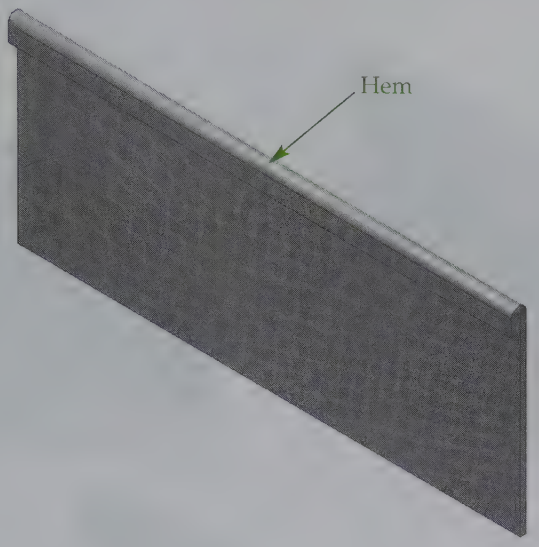
Placing Hems

A ***hem*** is a special type of flange typically used to add strength to or relieve sharpness of an exposed edge. See Figure 11-44. Hems also have multiple other functions, such as connecting sheet metal edges together. Hems do not require a sketch and are created using the **Hem** tool. Access the **Hem** tool by picking the **Hem** button on the **Sheet Metal Features** panel bar or picking the **Hem** button on the **Sheet Metal Features** toolbar. This opens the **Hem** dialog box displayed in Figure 11-45. To place a hem, first specify the type of hem you want to create by picking the type of hem from the **Type** drop-down list. A single hem is the default hem type, shown in Figure 11-44. All other hem type options are shown in Figure 11-46. Once you select the hem type, pick the **Select Edge** button—if it is not already selected, and pick the existing feature edge on which the hem will be created. The edge you select will identify the beginning of the hem width and the location of the bend.

Once you specify the type of hem you want to create and select the feature edge, you may need to pick the **Flip Direction** button, which allows you to identify the side of the existing feature you want the hem to be placed. Then, define the additional hem size and shape parameters using the options in the **Shape** area of the **Hem** dialog box for each of the following hem types:

- **Single and double hems.** When you create either a single or double hem, the **Gap** and the **Length** edit boxes are available. The gap represents the opening between the inside hem face and the inside feature face. By default, the gap is half of the material thickness specified in the **Sheet Metal Styles** dialog box. However, you can enter or select a different value in the **Gap** edit box. The length represents the amount of sheet metal material overlap used to create

Figure 11-44.
An example of a single sheet metal hem.



the hem. By default, the length is four times the material thickness specified in the **Sheet Metal Styles** dialog box. However, you can enter or select a different value in the **Length** edit box.

- **Teardrop and rolled hems.** When you create either a teardrop or a rolled hem, the **Radius** and the **Angle** edit boxes are available. The radius represents the radius of the bend. By default, the radius is equal to the bend radius specified in the **Sheet Metal Styles** dialog box. However, you can enter or select a different value in the **Radius** edit box. The angle represents the amount of sheet metal material overlap used to create the hem and the angle at which the material is bent. By default, the angle for a teardrop hem is 190°. However, you can enter any value greater than 180° and less than 359°, depending on the thickness of the material and the size of the radius. You also have the option of selecting a value in the **Angle** edit box.

Figure 11-45.
This is the **Hem** dialog box with the **Shape** tab displayed and the **Edge** extents option selected.

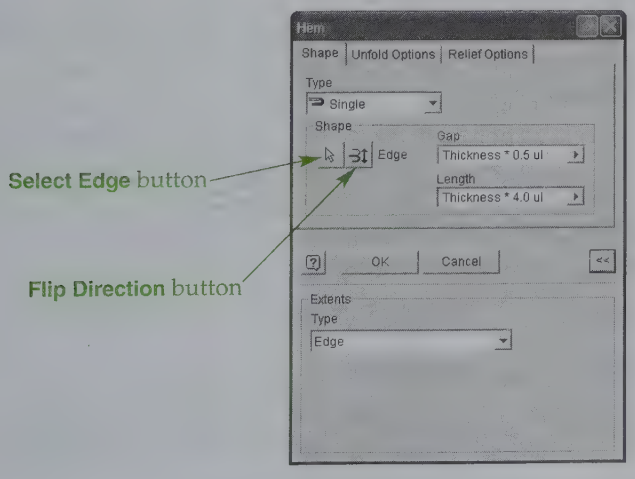
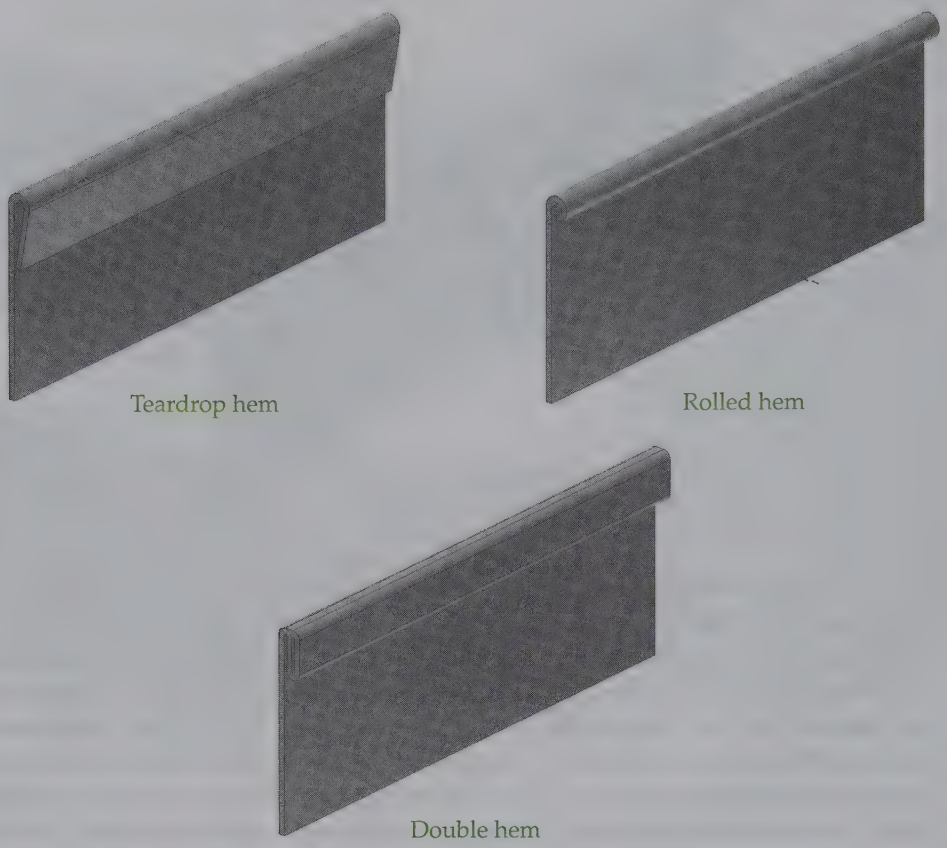


Figure 11-46.
An example of a teardrop hem, a rolled hem, and a double hem.



NOTE

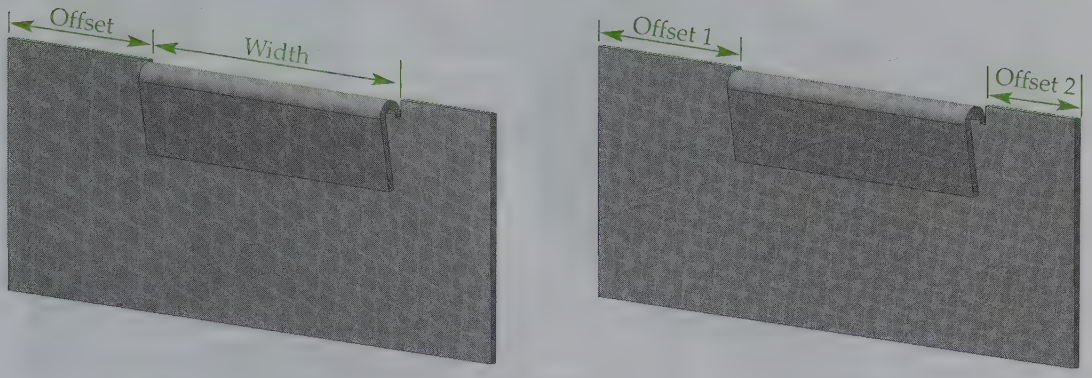
Entering hem gap, length, radius and angle values that are too small or large, or otherwise inappropriate for the part will result in a hem that intersects existing features, usually causing a modeling error.

Once you define all the hem shape options, you may want to modify the unfold and relief options located in the **Unfold Options** and **Relief Options** tabs of the **Hem** dialog box, as previously described in the sheet metal sketched feature tools.

The final step in creating a sheet metal hem is to define the hem extents, using the **Extents** area of the **Hem** dialog box, which is accessed by picking the **More** button. By default, the **Edge** extent type is active. This option, as shown in Figures 11-44 through 11-46, creates a hem that extends the entire length of the selected feature edge. In addition to the **Edge** extents option, the **Width** and **Offset** extents options are available in the **Hem** dialog box. These extents preferences function exactly the same as the **Width** and **Offset** extents options available in the **Flange** dialog box, as previously described. See Figure 11-47.

When you specify all the sheet metal hem options, pick the **OK** button if you are generating only one hem. If you want to use the **Hem** tool again, pick the **Apply** button and continue producing hems as needed. When finished, pick the **OK** button to exit the tool.

Figure 11-47.
Compare the dimensions required for the hem that uses the **Width** extents option to those of the **Offset** extents option.





Exercise 11-9 Complete the Exercise on the Student CD.

Creating Corner Seams

When creating sheet metal parts, it is often necessary to place seams at intersecting, perpendicular, or coplanar sheet metal faces. ***Corner seams*** add or remove material at the corners of sheet metal features to create a smooth transition and allow for unfolding. See Figure 11-48. Like other placed sheet metal features, corner seams do not require a sketch and can be added to any intersecting, perpendicular, or coplanar features such as two faces. Corner seams are created using the **Corner Seam** tool by picking the **Corner Seam** button on the **Sheet Metal Features** panel bar, or picking the **Corner Seam** button on the **Sheet Metal Features** toolbar. This opens the **Corner Seam** dialog box in Figure 11-49. The corner seam tool contains options that allow you to create many different corner seams. See the corner seam types and the details of the flat patterns used to build them in Figure 11-50.

To place a corner seam between faces that are not connected and are perpendicular to each other, pick the **Select Edge** button, if it is not already selected. Then pick one of the existing feature edges, followed by the other existing feature edge, to define the seam corner. Now, specify the type of corner seam you want to create by picking one of the three corner seam type buttons. A **No Overlap** corner seam is the default corner seam type; **Overlap** and **Reverse Overlap** are the other types. See Figure 11-51.

The next step in creating a corner seam is to define the gap between the selected sheet metal features by entering or selecting a value using the **Gap** edit box. The default gap value is the thickness of the material specified in the **Sheet Metal Styles** dialog box.

- A smooth transitioning corner seam between sheet metal faces that are not connected and are coplanar. To place a corner seam between faces that are not

Figure 11-48.
These are the folded and unfolded views of a non-overlapped corner seam applied to a face and two flanges with a round bend relief.

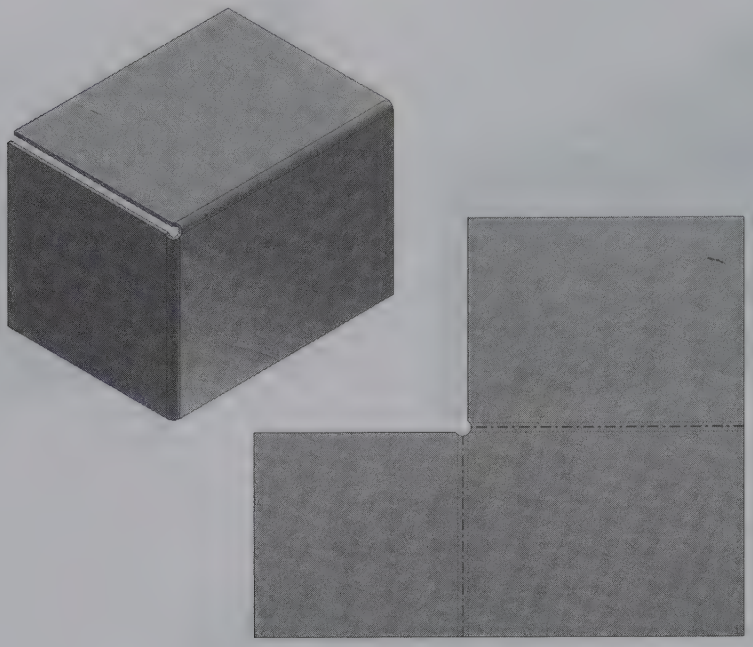
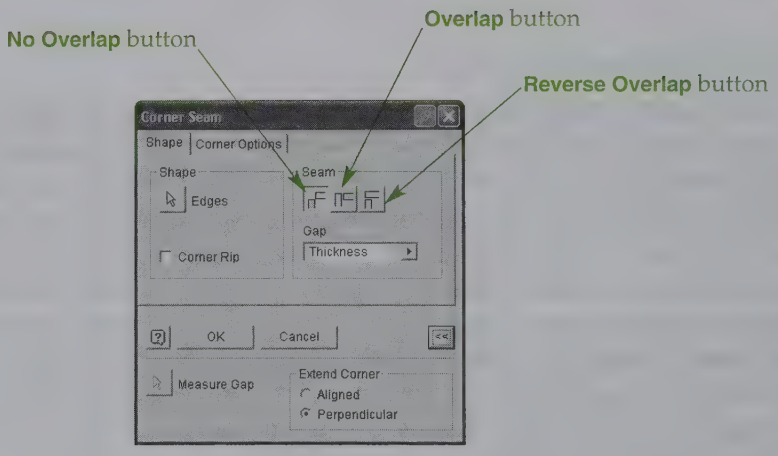


Figure 11-49.
The **Corner Seam** dialog box, **Shape** tab and **Seam** area.



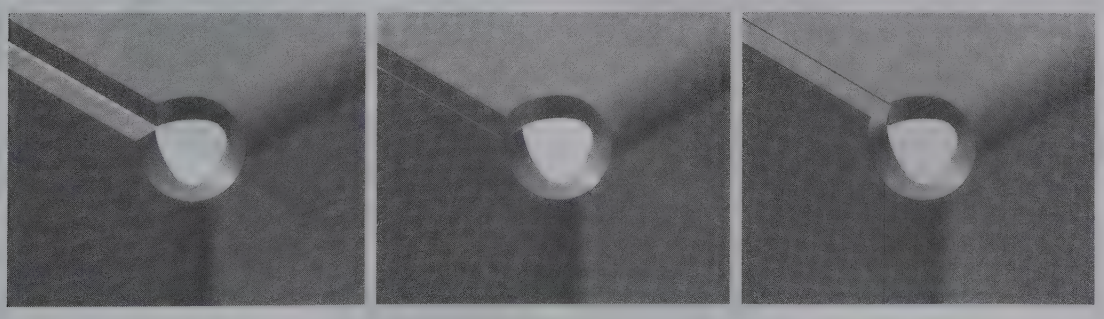
connected and are coplanar, as shown in Figure 11-52, pick the **Select Edge** button, if it is not already selected. Then pick an existing feature edge, followed by an edge on the coplanar face, perpendicular to the first edge selected, to define the location of the seam corner.

Once you select the feature edges, the **Seam** area of the **Corner Seam** dialog box changes to **Miter**. The **Miter** area allows you to specify the type of corner miter you want to create by picking one of the three corner miter type buttons. The effects of each of these options are shown in Figure 11-53. The next step in creating a corner seam is to define the gap between the selected sheet metal features by entering or selecting a value using the **Gap** edit box. The default gap value is the thickness of the material specified in the **Sheet Metal Styles** dialog box.

Figure 11-50.
The combinations of seam types and relief shapes provide many different corner styles.



Figure 11-51.
An example of an overlapped corner seam and a reverse overlap corner seam.



- A corner seam between sheet metal faces that are square and intersecting. A corner seam placed between intersecting square sheet metal features is known as a **corner rip**. To add a corner rip, as shown in Figure 11-54, ensure the **Select Edge** button is selected, pick the **Corner Rip** check box, and pick an existing square feature edge. Once you select the feature edge or edges, specify the type of corner rip you want

Figure 11-52.
Creating a corner seam between coplanar faces and the **Corner Seam** dialog box, **Shape** tab, and **Miter** area.

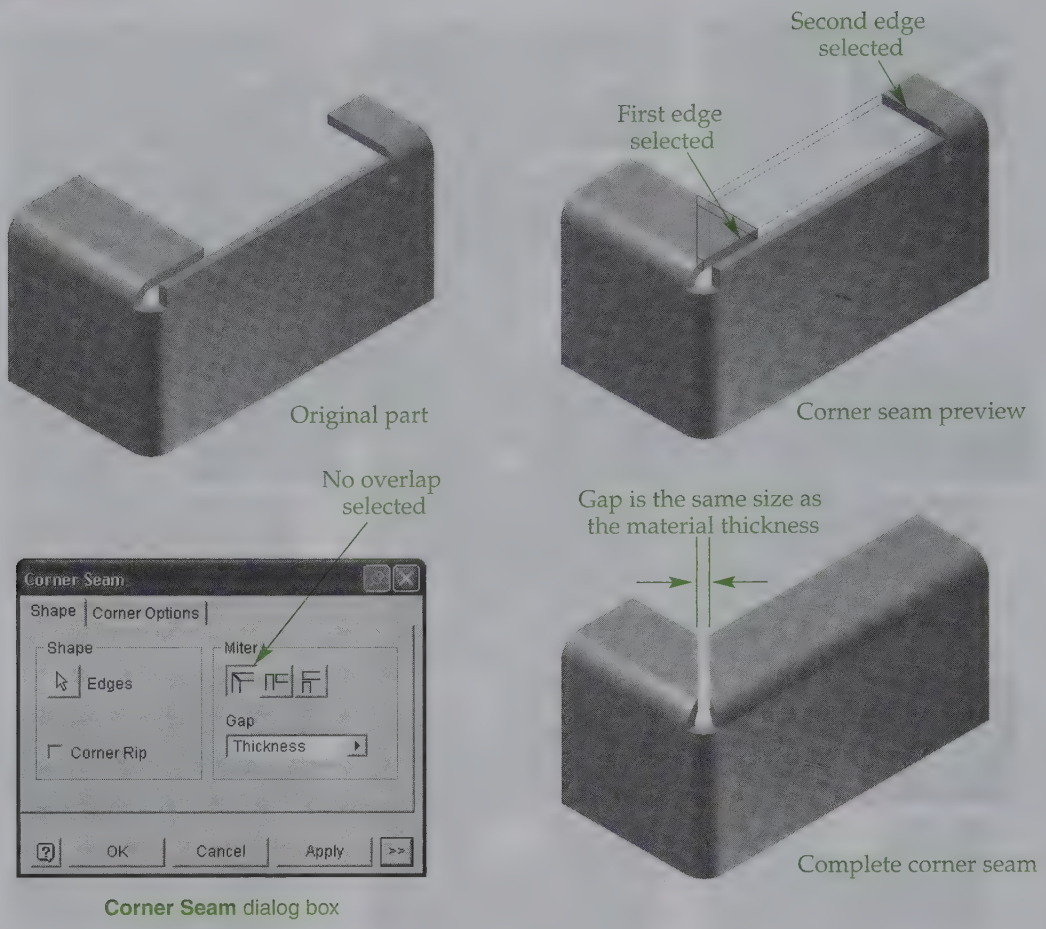
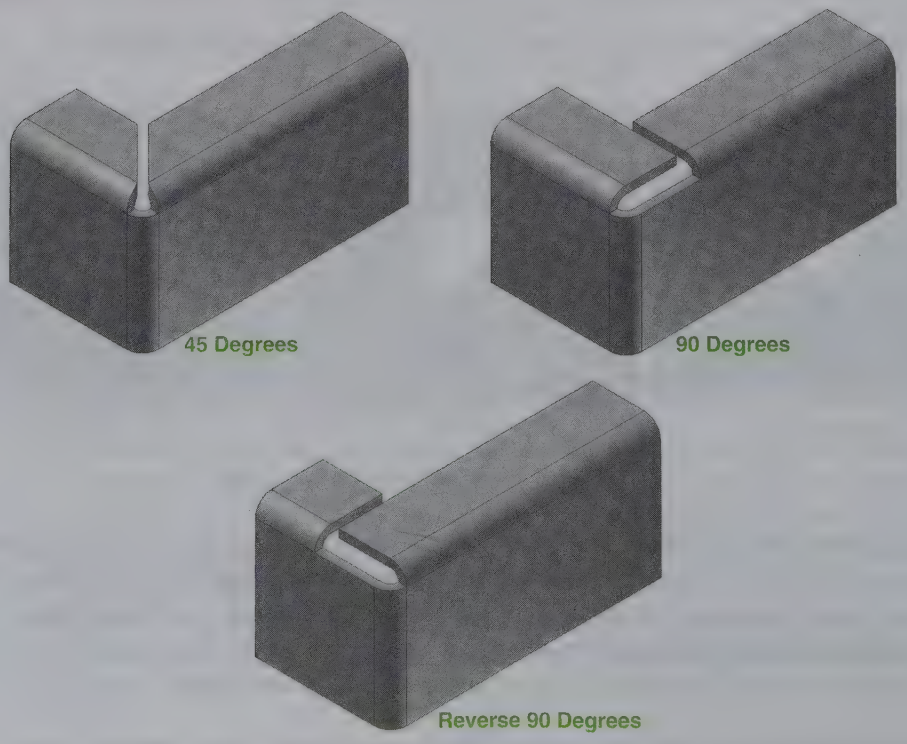


Figure 11-53.
An example of a **45 Degree** corner miter, a **90 Degree** corner miter, and a **Reverse 90 Degree** corner miter.



to create by picking one of the three corner seam type buttons. A **No Overlap** corner is the default corner seam type, shown in Figure 11-54. The other two corner rip types are shown in Figure 11-55.

Once you determine the type of corner seam you are creating and fully specify the corner seam shape options in the **Shape** tab of the **Corner Seam** dialog box, define the corner parameters using the **Corner Options** tab. The preferences in the **Corner Options** tab allow you to override the default options specified in the **Sheet Metal Styles** dialog box. However, if you want to use the sheet metal settings defined in the **Sheet Metal Styles** dialog box, select the **Default** option from the specified style parameter drop-down lists.

Finally, you can further define the corner seam using the following options accessed by picking the **More** button:

- **Measure Gap.** This button is available when you create a 45° corner miter.
- **Aligned.** This radio button aligns the selected face edges with each other by projecting the first face edge to the second face edge.
- **Perpendicular.** This radio button projects the first selected face edge perpendicular to the second selected face edge.

When you specify all the sheet metal corner seam options, pick the **OK** button if you are only generating one corner seam. If you want to use the **Corner Seam** tool again, pick the **Apply** button and continue producing corner seams as needed. When finished, pick the **OK** button to exit the tool.

NOTE

If you want to change a corner seam into a bend, right-click the corner seam in the browser and select the **Change to Bend** menu option. The corner seam is removed and the **Bend** dialog box is displayed.

Exercise 11-10 Complete the Exercise on the Student CD.

Figure 11-54.
Creating a corner rip.

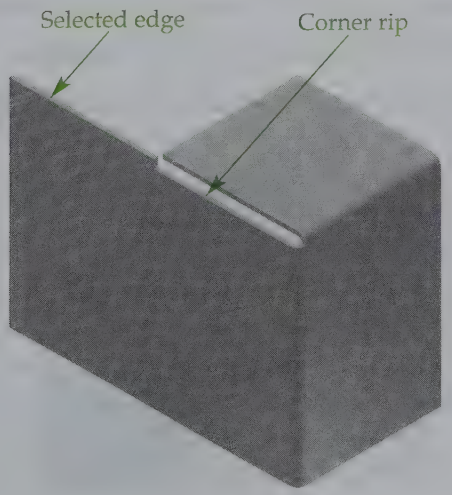
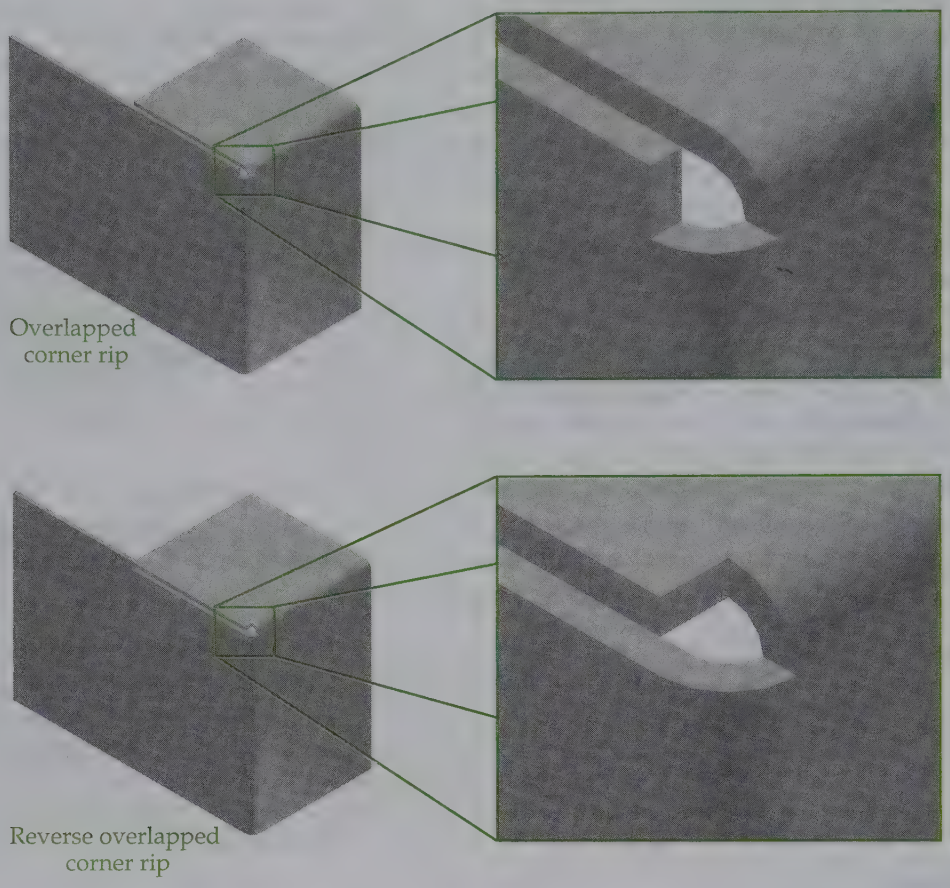


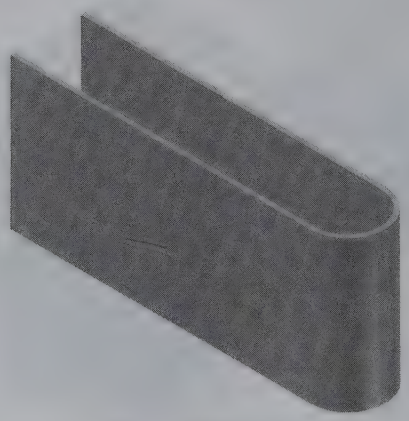
Figure 11-55.
An example of an overlapped corner rip and a reverse overlap corner rip.



Adding Bends

A sheet metal ***bend*** is typically created during sheet metal part development using the tools previously described. However, sometimes it is necessary to add an additional bend between two perpendicular, or two parallel, sheet metal features. See **Figure 11-56**. Once you create at least two acceptable sheet metal features, access the **Bend** tool by picking the **Bend** button on the **Sheet Metal Features** panel bar, or picking

Figure 11-56.
An example of a sheet metal bend between two parallel sketched faces, full radius bend.

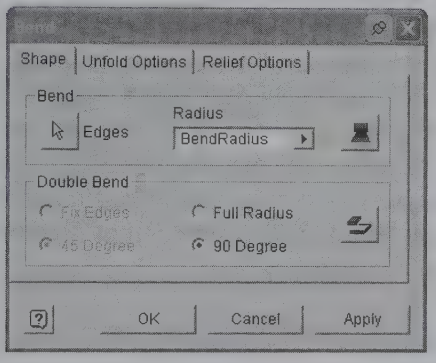


the **Bend** button on the **Sheet Metal Features** toolbar. This displays the **Bend** dialog box shown in Figure 11-57.

To create a sheet metal bend, pick the **Select Edges** button from the **Bend** area of the **Shape** tab, if it is not already active. Then pick an edge on both features where you want the bend to be created. When both features are selected, a preview of the bend is displayed. The next step is to define the face bend using the options found in the **Shape** tab. The **Radius**, **Extend Bend Aligned to Side Faces**, **Double Bend**, and **Flip Fixed Edge** options available in the **Bend** dialog box, function the same as those found in the **Face** dialog box.

Once you define all the bend shape options, you may want to modify the **Unfold** and **Bend Relief** options located in the **Unfold Options** and **Relief Options** tabs of the **Bend** dialog box, as previously described in the sheet metal sketch tools section of this chapter. Finally, when you specify all the sheet metal bend options, pick the **OK** button if you are generating only one bend. If you want to use the **Bend** tool again, pick the **Apply** button and continue producing bends as needed. When finished, pick the **OK** button to exit the tool.

Figure 11-57.
The **Bend** dialog box, **Shape** tab.



NOTE

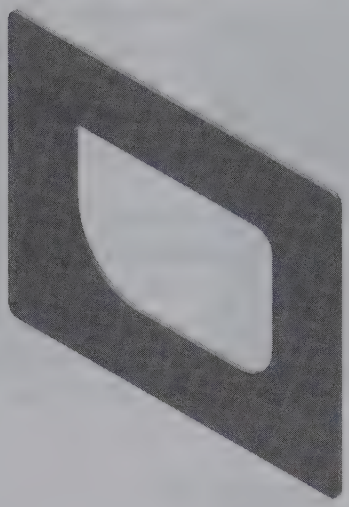
If you want to change a bend into a corner seam, right-click the bend in the browser and select the **Change to Corner** menu option. The bend is removed, and the **Corner Seam** dialog box is displayed.

Exercise 11-11 Complete the Exercise on the Student CD.

Placing Corner Rounds

A ***corner round*** in the sheet metal environment is very similar to a fillet or round in the part model environment. Corner rounds are curves placed at inside and outside sheet metal corner intersections. See Figure 11-58. Typically, corner rounds are used to remove sharp points and edges, or to apply a required radius. Corner rounds can be placed at the corner of any acceptable sheet metal feature, and are created using the **Corner Round** tool. Access the **Corner Round** tool by picking the **Corner Round** button

Figure 11-58.
Examples of sheet metal corner rounds.



on the **Sheet Metal Features** panel bar, or picking the **Corner Round** button on the **Sheet Metal Features** toolbar. This opens the **Corner Round** dialog box in Figure 11-59. Most of the time, you should be able to apply all the corner rounds needed in the part using a single **Corner Round** tool, because the **Corner Round** dialog box allows you apply multiple corner rounds with different radii. You do not need to access the **Corner Round** tool continually to create a different type of corner round. Figure 11-58 shows an example of a sheet metal part with a variety of corner round radii. All the fillets and rounds were created using a single **Corner Round** tool, and each of the corner rounds placed during one operation create a single feature.

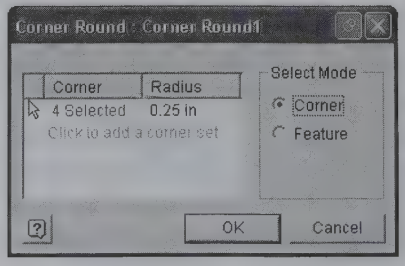
PROFESSIONAL TIP



If you plan to suppress only certain corner rounds or want to have additional corner rounds available, you may want to access the **Corner Round** tool more than once.

To apply a corner round to a square sheet metal feature corner, first specify how you want to select the corners, by picking either the **Corner** or the **Feature** radio button. By default, the **Corner** radio button is selected. This option allows you to select specific corners individually. If you want to apply corner rounds to all the corners on an entire feature, pick the **Feature** radio button. Once you determine how you want to

Figure 11-59.
The **Corner Round** dialog box.



select corners to apply corner rounds, specify a corner round radius by highlighting the default radius and entering the desired value, or by selecting a value.

The next step is to pick the corners where you want to apply corner rounds. If you are using the **Corner** select method, identify to which corners you want to apply rounds by selecting the individual corners. If you are using the **Feature** select method, identify the features to which you want to apply corner rounds by selecting entire features. A preview of the selected corners is displayed. If the preview looks correct, and you do not want to place additional corner rounds with different radii, pick the **OK** button to generate the curves. If you want to apply other corner rounds with different radii, pick the **Click to add a corner set** button. Then, enter the corner round radius and select corners. Continue adding corner rounds as required. When finished, pick the **OK** button.

NOTE

You can deselect any of the chosen corners by holding down [Ctrl] and picking the object. To delete a corner round corner listed in the **Corner** list box, pick the corner you want to remove and press the [Delete] key on your keyboard.

Exercise 11-12 Complete the Exercise on the Student CD.

Adding Corner Chamfers

A ***corner chamfer*** is an angled face that replaces a square corner on a sheet metal feature. See Figure 11-60. Corner chamfers are used to relieve sharp edges and points or create specific angled sheet metal geometry. Corner chamfers can remove material from an external corner or add material to an internal corner, and are created using the **Corner Chamfer** tool. Access the **Corner Chamfer** tool by picking the **Corner Chamfer** button on the **Sheet Metal Features** panel bar, or picking the **Corner Chamfer** button on the **Sheet Metal Features** toolbar. This opens the **Corner Chamfer** dialog box.

NOTE

You need to access the **Corner Chamfer** tool every time you want to create a different-size chamfer. However, you can fully chamfer a sheet metal part using only one **Corner Chamfer** tool if the distances are constant for each corner. Still, each of the corner chamfers placed during one operation create a single feature.

To create a corner chamfer, first select one of the following corner chamfer method buttons: **One Distance**, **Distance and Angle**, and **Two Distances**. The techniques used by each of these buttons to create chamfers are shown in Figure 11-61. The method button selected by default is **One Distance**, and is the simplest option to use. The **One Distance** option allows you to create a corner chamfer with equal distance sides from the selected corner, or a 45° angle. To place a corner chamfer using the **One Distance** option, you must first be sure the **One Distance** button is selected. Then pick the edges you want to chamfer, and enter or select a corner chamfer distance using the **Distance** edit box. If the chamfer preview looks correct, pick the **OK** button to create the chamfer.

Figure 11-60.
Examples of several chamfers placed on a part.

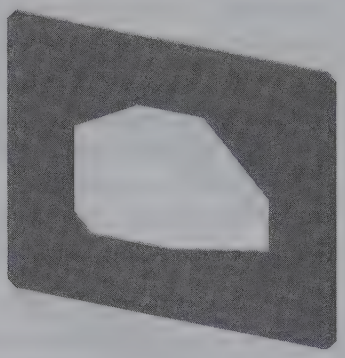
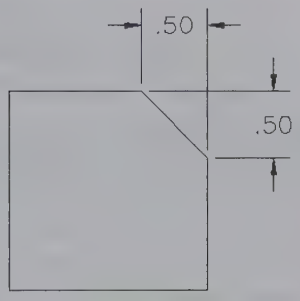
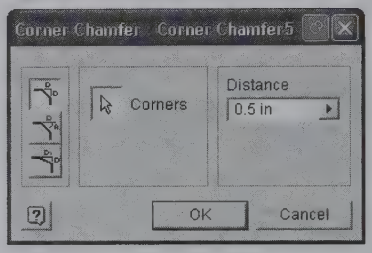
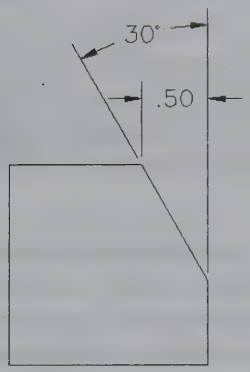


Figure 11-61.
There are three different methods of defining chamfers.

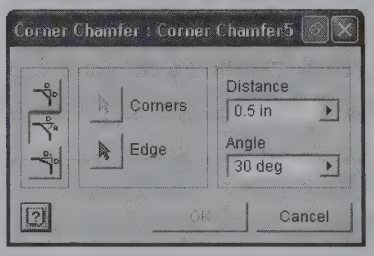


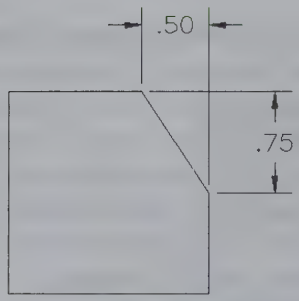
One Distance



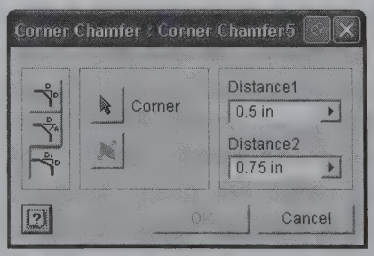


Distance and Angle





Two Distance



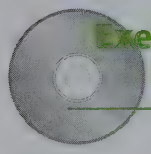
NOTE



You can deselect any of the selected by holding down [Ctrl] and picking the corner.

If you want to specify one corner chamfer side as a distance from the corner, and the other side as an angle from the corner, pick the **Distance and Angle** method button. Initially, the **Edge** button should be active. The **Edge** button allows you to select the face you want the distance from the corner to be measured. Once you define the face, the **Corners** button should become active. The **Corners** button allows you to select the corners from where you want the angle to be measured. To change the face, pick the **Edge** button again, and then reselect the corners. Once you establish the face and corners you want to chamfer, enter or select a corner chamfer distance using the **Distance** edit box, and enter or select a corner chamfer angle using the **Angle** edit box. If the chamfer preview looks correct, pick the **OK** button to create the chamfer.

To specify one corner chamfer side with a distance from the edge, and the other side with a different distance from the edge, pick the **Two Distances** method button. To place a corner chamfer using the **Two Distances** method, with the **Corner** button selected, pick the corners you want to chamfer. Then enter or select a chamfer distance using the **Distance1** edit box, and enter or select a chamfer distance using the **Distance2** edit box. If the corner chamfer preview looks correct, pick the **OK** button to create the corner chamfer. However, if the distances you enter need to be reversed, pick the **Flip** button to redefine the direction of the corner chamfer distances from the edge, and pick the **OK** button to create the corner chamfer.

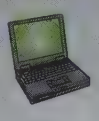


Complete the Exercise on the Student CD.

Using the PunchTool

The Inventor sheet metal PunchTool is used to place special sheet metal iFeatures on a sheet metal part. Using the PunchTool is very similar to inserting an iFeature in the part environment. However, the base feature sketch used to create the iFeature must contain a **Point, Hole Center**, because a **PunchTool** iFeature must be inserted on a sketched **Point, Hole Center**. See Figure 11-62. Once you create an iFeature with a sketched **Point, Hole Center** for positioning and have placed **Point, Hole Centers** at the desired locations on the sheet metal feature, you are ready to use the sheet metal PunchTool to insert or "punch" features.

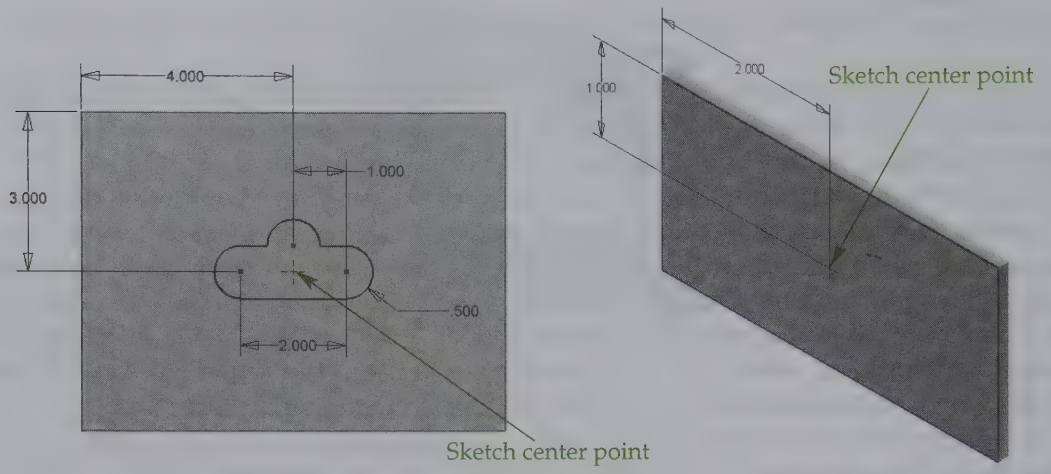
PROFESSIONAL TIP



You may want to save all your PunchTool iFeatures in a specific location. The Inventor PunchTool iFeatures are located in the following path: Program Files/Autodesk/Inventor 11/Catalog/Punches.

Access the **PunchTool** tool to open the **PunchTool** dialog box, shown in Figure 11-63, by picking the **PunchTool** button on the **Sheet Metal Features** panel bar, or picking the

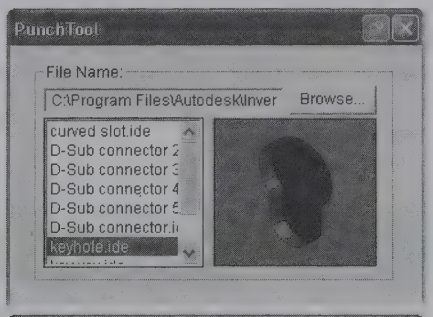
Figure 11-62.
Sketch center points are required for the base feature sketch of the **PunchTool** iFeature and the sheet metal feature receiving a **PunchTool** iFeature.



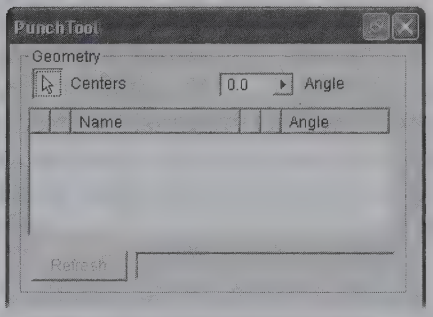
The base feature sketch of the **PunchTool** iFeature

Sheet metal feature

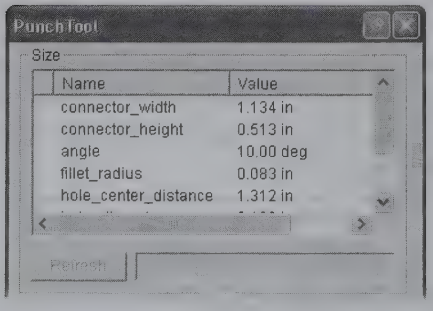
Figure 11-63
The **PunchTool** dialog box changes as you progress through the steps of adding sheet metal iFeatures.



The **PunchTool** dialog box, select file window contains a list of available punches and a preview of the selected punch



The position geometry window is used to add one or more punches to the selected part and define the angle of the punches



The modify size parameters window allows you to change the size of certain parameters

PunchTool button on the **Sheet Metal Features** toolbar.

When you initially open the **PunchTool** dialog box, the select file window is displayed, and the contents of the **Punches** folder is shown in the file list box. If you want to use one of the iFeatures found in the **Punches** folder, select the file from the list box and pick the **Next** button. If you want to locate a different folder that contains other **PunchTool** iFeatures, pick the **Browse...** button to access the **Browse For Folder** dialog box. Here you can locate and select any folder on your computer or the network. When you find the desired folder, pick the **OK** button. The folder path should be displayed in the **File Name** area, and the files contained in the folder should be listed in the file list box. Select the iFeature file you want to use, and pick the **Next** button.

When you select the **Next** button, the position geometry window should be displayed. In addition, the selected iFeature is automatically positioned on the sketched **Point, Hole Centers**, and a preview of the design element is displayed. If you need to select additional hole centers, pick the **Select Centers** button, if it is not already active, and pick the hole centers. However, if you want to deselect certain hole centers, and not place the iFeature at those locations, ensure the **Select Centers** button is selected, hold down the [Ctrl] key on your keyboard, and pick the hole centers you want to deselect. The next step is to define the angle of the punches by entering or selecting a value in the **Angle** edit box.

The final step in positioning the iFeature is to satisfy any dangling geometry references. ***Dangling geometry*** results when additional positioning information is required in order for iFeature insertion to occur, and is primarily due to issues with the initial iFeature sketch and existing feature geometry. Dangling geometry references may include faces, edges, angles, or other reference points, and are displayed in the **Dangling Geometry** list box. To satisfy dangling geometry, pick the name in the list box, and follow the prompt in the position geometry window of the **PunchTool** dialog box. After you position the iFeature, pick the **Next** button to activate the **Modify Size Parameters** window.

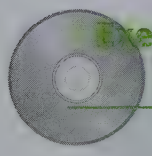
If you specified specific size parameter information when the iFeature was created, the size parameters are listed in the **Size** list box; otherwise the area is blank. The prompts guide you through the process of sizing the iFeature. If no size limits were placed on the iFeature when it was created, you can enter any value. However, if a range limit was placed on the iFeature when created, you will be able to enter only a specific range of values, or if a list limit was placed on the iFeature when created, you will be able to choose only values from the **Value** drop-down list.

NOTE



Pick the **Refresh** button in the **PunchTool** dialog box any time you resize or reposition the iFeature, to preview the changes. Pick the **Back** button in the **PunchTool** dialog box any time to return to the previous window to modify an insertion option.

Once you select, position, and size the PunchTool iFeature you want to insert, pick the **OK** button to insert the iFeature. However, if you at least select and position the iFeature, you can choose the **OK** button and bypass the size options.



Complete the Exercise on the Student CD.

Creating Flat Patterns

Once you create a sheet metal part, or any time during sheet metal part development, you can unfold the entire part and create a *flat pattern*. A flat pattern is the flat layout of the sheet metal part, which can eventually be transferred to a flat piece of sheet metal and formed into the sheet metal part. See Figure 11-64. The **Flat Pattern** tool is used to create a flat pattern from an Inventor sheet metal part. Access the **Flat Pattern** tool by picking the **Flat Pattern** button on the **Sheet Metal Features** panel bar, or picking the **Flat Pattern** button on the **Sheet Metal Features** toolbar.

NOTE



In order to effectively view bend center lines and extents while in a shaded display mode, the **Active Edge Display** check box must be selected from the **Display** tab of the **Options** dialog box.

The **Flat Pattern** tool calculates all the features and bends in the part and creates a flat pattern. You can create a flat pattern from any sheet metal part. If the sheet metal part contains at least one bend, when you access the **Flat Pattern** tool the part is automatically flattened. However, if the part does not contain any bends, such as a cylindrical or conical sheet metal feature, you must first select a face. You can also select any face, on any part, to define that face as the flat pattern base. When you generate a flat pattern, the **FlatPattern** specification and icon are displayed in the browser, as shown in Figure 11-65, and a new window is automatically created with the flat pattern displayed in the graphics window. You cannot modify the sheet metal part in the flat pattern window. However, changes made to the sheet metal part, in the ipt. file, are updated in the flat pattern, unless the change made in the sheet metal part cannot be flattened.

NOTE



Typically, the parent part file is hidden by the new flat pattern, but is not closed. You can arrange the part file and flat pattern windows just like any other windows. The flat pattern is differentiated from the parent sheet metal part as **FlatPattern**, followed by the name of the part.

Figure 11-64.
An example of a sheet metal flat pattern and the formed part.

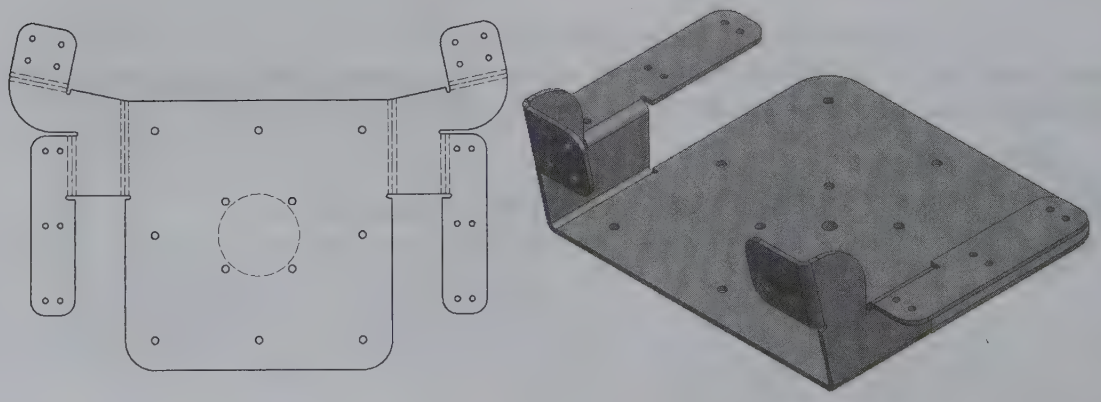
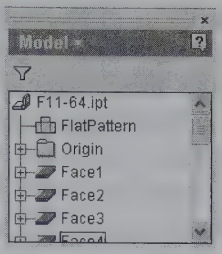


Figure 11-65.
The **FlatPattern** specification and icon in the browser.



Once you create a flat pattern, the following menu options specific to flat patterns are available when you right-click on **FlatPattern**, or the flat pattern icon in the browser:

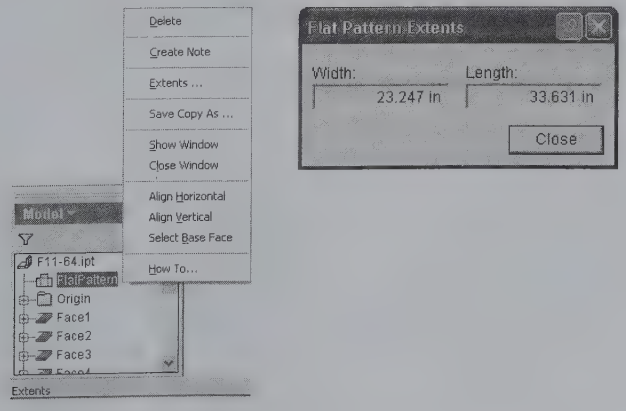
- **Extents...** This option opens the **Flat Pattern Extents** display box, which allows you to view the overall length and width of the piece of sheet metal material needed to create the part. See Figure 11-66.

NOTE

You cannot edit the length and width in the **Flat Pattern Extents** display box, but you can use measuring tools to view pattern measurements.

- **Save Copy As...** This option opens the **Save Copy As** dialog box, which allows you to save a copy of the flat pattern as an SAT, DWG, or DXF file, anywhere on your computer or the network.
- **Show Window.** If the flat pattern graphics window is not active and is hidden by another window, pick this option to display the flat pattern graphics window.
- **Close Window.** Use this option to close the flat pattern graphics window.
- **Open Window.** This option allows you to reopen the flat pattern graphics window and is only available when the flat pattern window has been closed.
- **Align Horizontal.** Use this option to horizontally align, or revolve, the flat pattern in reference to an edge on the sheet metal part. When you select the **Align Horizontal** option, the sheet metal part file used to generate the flat pattern automatically opens, and you are required to select an edge. The edge you pick is used to reposition and horizontally align the flat pattern. You must then

Figure 11-66.
Right-click the **FlatPattern** icon and pick **Extents...** to open the **Flat Pattern Extents** dialog box.



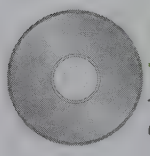
reopen the flat pattern window to observe the display change.

- **Align Vertical.** Use this option to vertically align, or revolve, the flat pattern in reference to an edge on the sheet metal part. When you select the **Align Vertical** option, the sheet metal part file used to generate the flat pattern automatically opens, and you are required to select an edge. The edge you pick is used to reposition and vertically align the flat pattern. You must then reopen the flat pattern window to observe the display change.
- **Select Base Face.** This option allows you to reselect the flat pattern base feature from the sheet metal part. To select a base feature, pick the **Select Base Face** menu option and choose the feature you want to specify as the base feature.

NOTE



Often complex features, such as embossed iFeatures, cannot be flattened, and remain three-dimensional when other features are flattened.



Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. Describe *sheet metal* parts and their use.
2. When can sheet metal sketched features be created and give an example of a sheet metal sketched feature?
3. What is a *sheet metal face* and where is it created by default?
4. Give the function of a *corner relief.*
5. Briefly describe a *sheet metal contour flange.*
6. Give the function of the sheet metal **Cut** tool.
7. Describe the basic reason for creating a *sheet metal fold.*
8. What is usually used to define the fold edge of a sheet metal fold?
9. What must be present before you can add sheet metal placed features?
10. When can sheet metal placed features be created, and give an example of at least two sheet metal placed features?
11. Define *flange.*
12. Give a brief definition of *hem.*
13. Explain the basic purpose of *corner seams.*
14. What is a corner seam placed between intersecting square sheet metal features called?
15. When is a sheet metal bend created?
16. Define *corner round* and its use.
17. Define *corner chamfer* and its use.
18. Explain the use of the Inventor sheet metal **PunchTool.**
19. Define *dangling geometry.*
20. What is a *flat pattern* and when can it be created?

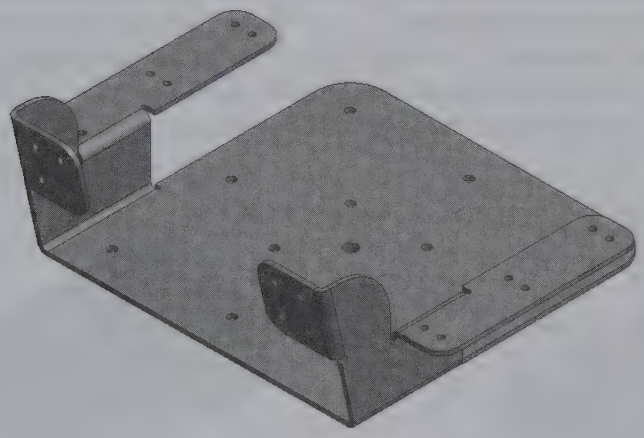
Problems

Instructions:

- Follow the specific instructions for each problem.

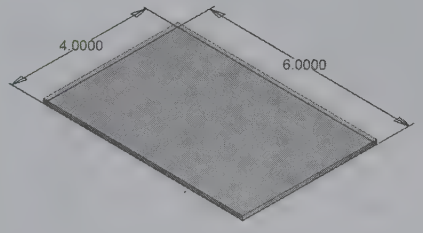
1. **Name:** Seat Base
   Units: Inch or Metric
   Template: SHEET METAL-in.ipt or SHEET METAL-mm.ipt
   Material: Default
   Color: Default
   Sheet Metal Style: Custom
   Specific Instructions:
   - Using the part modeling techniques described in previous chapters and the sheet metal part modeling techniques described in this chapter, create a seat base similar to the seat base shown.
   - When complete, use the flat pattern tool to view the seat base flat pattern.

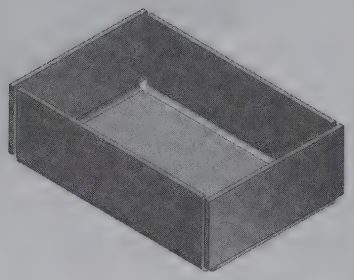
   Save the sheet metal part as: P11-1.ipt.
   Save the flat pattern as: P11-1.sat.

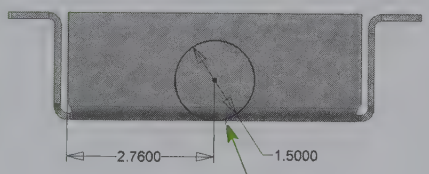


2. **Name:** Housing
Units: Inch
Template: SHEET METAL-in.ipt
Material: Default
Color: Default
Sheet Metal Style: Default
Specific Instructions:
- Begin a new inch sheet metal part file.
- Create a 4″ × 6″ face as shown. Use default sheet metal style settings throughout this problem.
- Add 2″ flanges around the face as shown. The flanges should offset toward the face and occur in a positive direction. Add two flanges to the 4″ long sides first.
- Add two 1″ flanges as shown. The flanges should offset toward the base flanges.
- Place four .25″ holes on the flanges as shown.
- Modify each of the corners with a **No Overlap** seam and a round bend relief as shown.
- Sketch and constrain a circle on the face shown, and cut the circle across the bend of the specified flange.
- Sketch and constrain a rectangle on the face shown, and cut the rectangle across the bend of the specified flange.

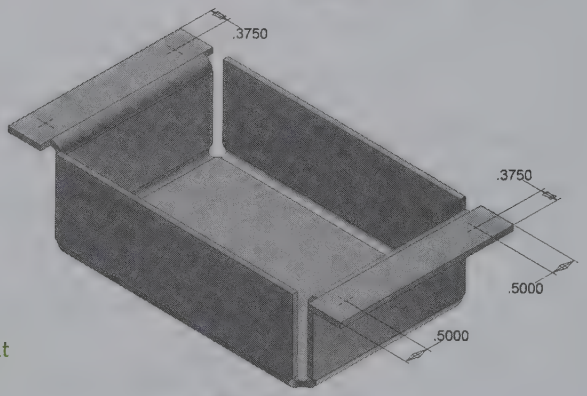
The final part should look like the part shown.
Save As: P11-2.ipt.

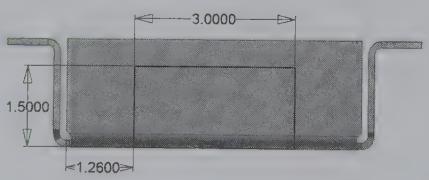


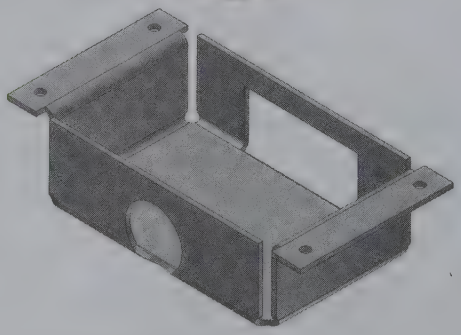




Tangent Constraint

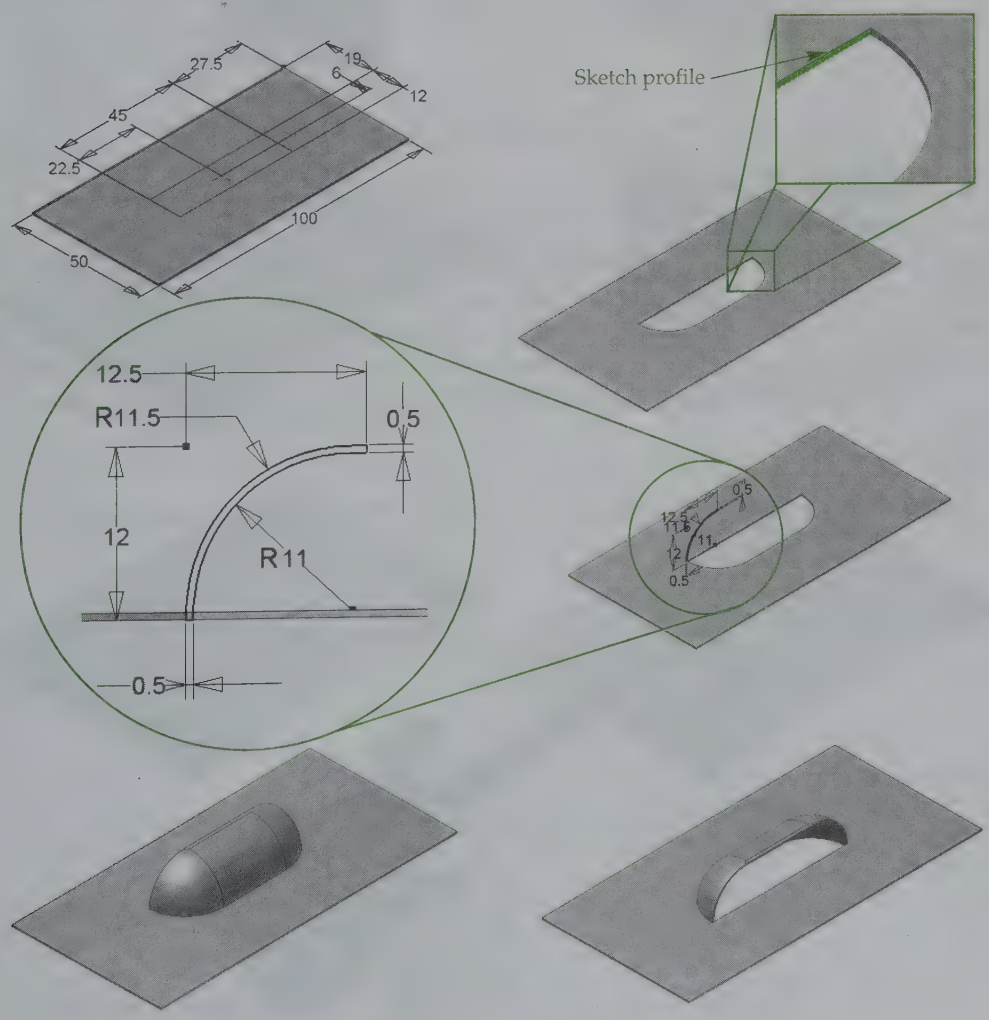






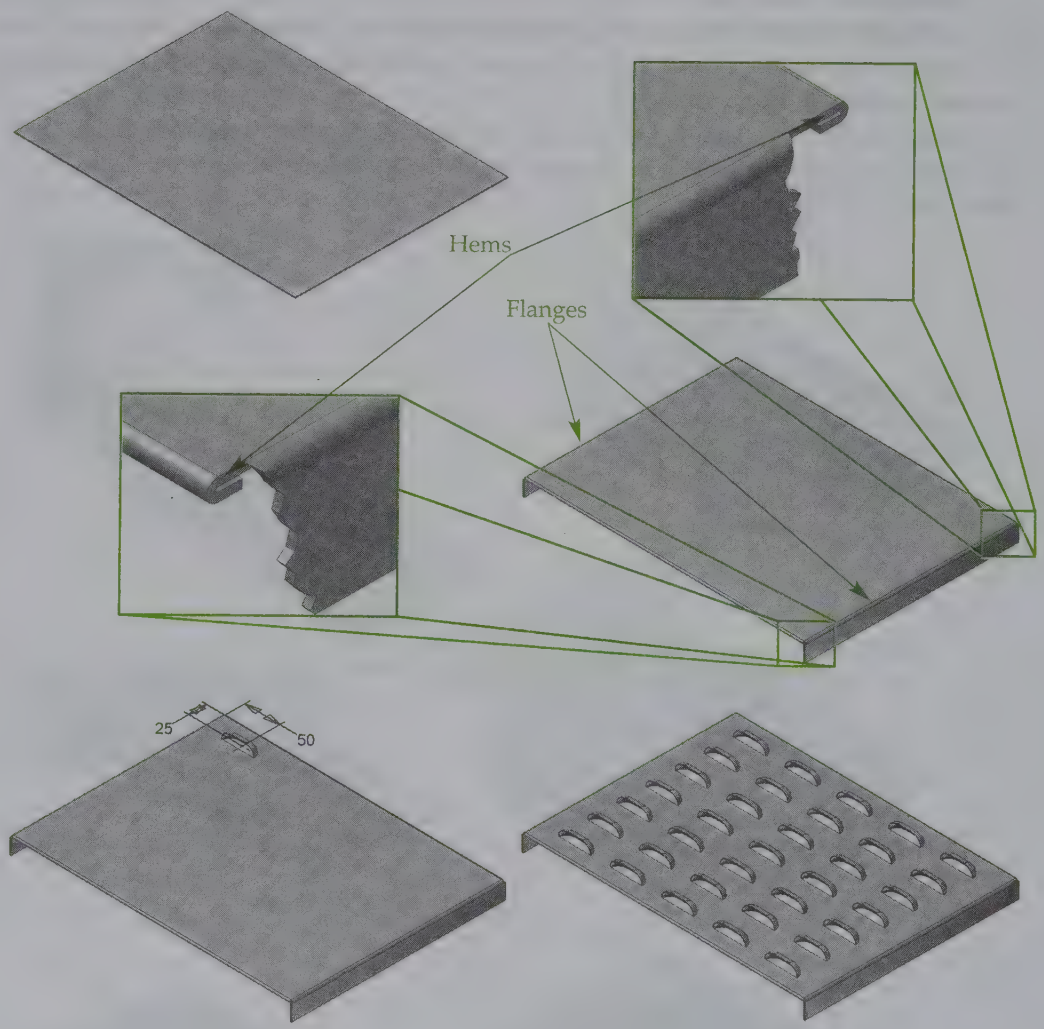
3. **Name:** Louver
Units: Metric
Template: SHEET METAL-mm.ipt
Material: Default
Color: Default
Sheet Metal Style: Default
Specific Instructions:
- Begin a new metric sheet metal part file.
- Create a 50 mm × 100 mm face as shown.
- Sketch the rectangle and **Point, Hole Center** on the face as shown for reference.
- Cut the rectangle through the face.
- Place R 12 mm corner rounds as shown.
- Open a sketch on the inside of the cut and sketch the geometry shown. You may want to use **Slice Graphics** for clarity. This sketch is used as a sweep profile.
- Open a new sketch on the top of the face and project the geometry shown. This sketch is used as a sweep path.
- Access the **Part Features** tools, and create a sweep feature using Sketch3 as the profile and Sketch4 as the path. The sweep should look like the feature shown.
- Using the **Create iFeature** dialog box, create an iFeature from the cut, corner round, and sweep. Name the iFeature Louver, and specify thickness, length, and width size parameters.
- Save the iFeature in a location of your choice for future access.

The final part should look like the part shown.
Save As: P11-3.ipt.



4. **Name:** Louvered Cover
Units: Metric
Template: SHEET METAL-mm.ipt
Material: Default
Color: Default
Sheet Metal Style: Change the default sheet metal style thickness to 1.5 mm.
Specific Instructions:
- Begin a new metric sheet metal part file.
- Create a 300 mm × 400 mm face as shown.
- Add 25 mm flanges to the 300 mm edge as shown.
- Add single hems to the 400 mm edge as shown.
- Sketch a **Point, Hole Center** on the top of the face as shown.
- Insert the louver iFeature created in P11-3.ipt on the **Point, Hole Center**.
- Use the **Rectangular Pattern** tool, and equally space louvers as shown.

The final part should look like the part shown.
Save As: P11-4.ipt.



CHAPTER 12

Assemblies

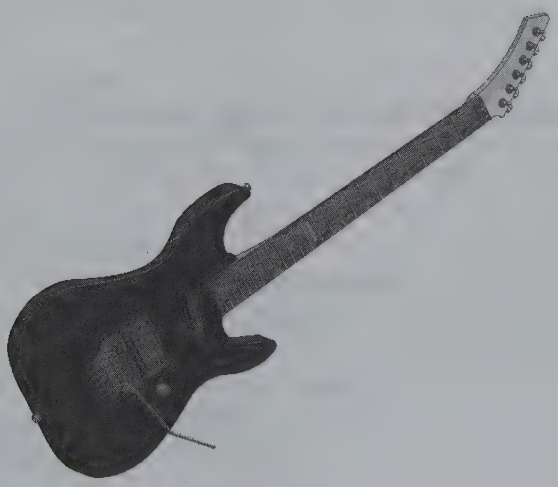
Learning Objectives

After completing this chapter, you will be able to do the following:

- Create assemblies using existing components.
- Develop assemblies in place.
- Place, edit, and drive assembly constraints.
- Pattern assembly components.
- Mirror and copy assembly components.
- Adjust assemblies.
- Apply assembly section views.
- Work with assembly design view and positional representations.

An ***assembly***, as shown in **Figure 12-1**, is a grouping of one or more design components. Components may include part models and subassemblies. A ***subassembly*** is an assembly that is placed in a larger assembly. For example, subassemblies may include switches or spring assemblies, which are composed of two or more parts, and may be used more than once in the final assembled product. In an Inventor assembly file, multiple parts and subassemblies are brought together and constrained to form an assembly. Constraints in the assembly environment are used to link individual components together.

Figure 12-1.
An example of a guitar assembly, which consists of numerous parts and subassemblies.



Developing Assembly Models

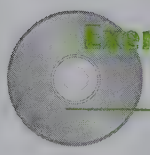
Inventor assembly files are separate files that carry the extension .iam. There are two main options for developing assemblies and subassemblies. One method involves inserting existing parts and subassemblies into an assembly file. The second technique involves creating individual components in an assembly file without inserting existing components. For this method, you may find it helpful to think of an assembly file as a part or subassembly file that also contains assembly tools. Although either option is effective, it is often faster, easier, and more productive to develop components within an assembly file, because building parts and subassemblies inside an assembly file creates both an assembly and a separate part or assembly file for each component. Often, a combination of inserting existing components and producing components in-place is used when creating assemblies.

By default, when you open a new Inventor assembly file, the work environment should look like the display in Figure 12-2. When you open an assembly file, depending on your approach, you are ready to insert components and create components in-place using assembly tools. Assembly tools are used to develop assemblies by inserting parts and subassemblies, creating components, placing assembly constraints, patterning components, and various other processes. Assembly tools are accessed from the **Assembly Panel** toolbar, the **Assembly Panel** bar, or specific assembly toolbars. In addition, many of the tools can be accessed from the **Edit**, **Insert**, and **Tools** pull-down menus.

NOTE



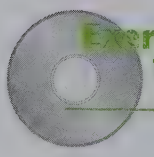
The feature tools available from the **Assembly Panel** toolbar, **Assembly Panel** bar, or specific assembly toolbars are primarily used for post-assembly operations, such as drilling a hole through two or more components, or any other machining processes. Do not confuse these feature tools with those used to create part or subassembly models, even though they function exactly the same. Using feature tools to modify assemblies does not affect individual part or subassembly models outside of the parent assembly file.



Complete the Exercise on the Student CD.

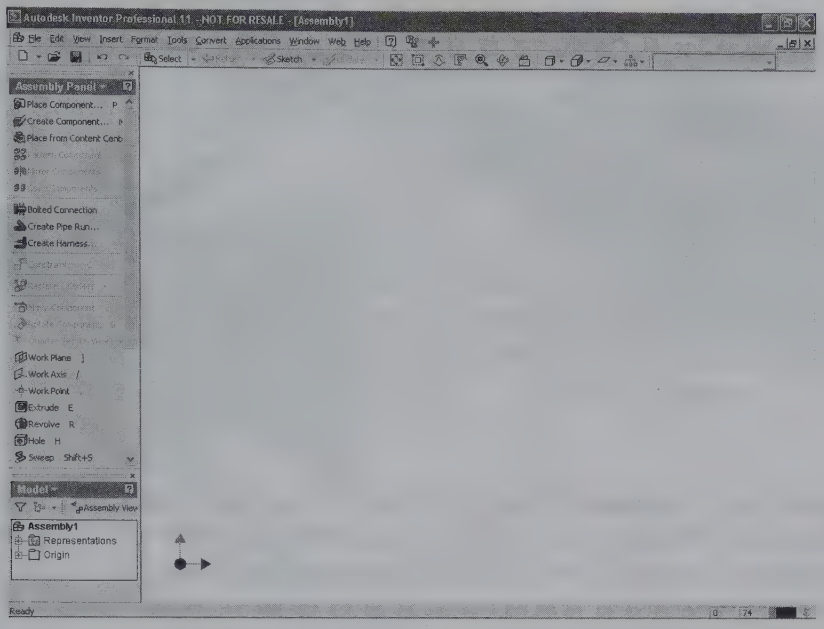


Complete the Exercise on the Student CD.



Complete the Exercise on the Student CD.

Figure 12-2.
The default Inventor assembly work environment.



Creating Assemblies Using Existing Components

One option for generating an assembly is to insert existing parts or subassemblies into an assembly file. Typically, the first component you insert should be a major part or subassembly, such as the assembly body or a main support base. The initial part or subassembly, which is the first component listed in the browser, is similar to the base feature of a part. However, defining the first component is usually not as critical as specifying a base feature. When you insert the initial assembly component, it automatically becomes grounded. As the name implies, a ***grounded*** part or assembly cannot be moved or shifted because it is fastened to a point, or to three-dimensional space. The initial component is grounded to the assembly origin, or center point, located in the browser **Origin** folder. Each component you insert after the first component is not grounded and can be placed anywhere in the graphics window.

NOTE

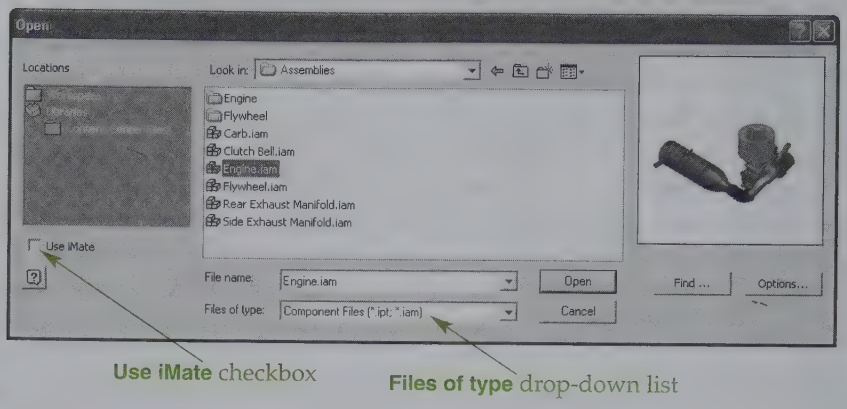


A ground constraint can be removed by right-clicking the component name in the browser, or the actual component in the graphics window, and deselecting the **Grounded** menu option.

If you choose to apply the method of inserting existing components to create an assembly, the first tool you use is the **Place Component** tool that accesses the **Open** dialog box in Figure 12-3, used to insert parts or subassemblies into an Inventor assembly file. Access the **Place Component** tool by picking the **Place Component** button on the **Assembly Panel** bar; picking the **Place Component** button on the **Assembly Panel** toolbar; pressing [P]; selecting the **Place Component...** option from the **Insert** pull-down menu; or right-clicking and selecting the **Place Component** pop-up menu option.

The main difference between the **Open** dialog box displayed when placing an assembly component and other **Open** dialog boxes you have used for other applications is the **Files of type** drop-down list, which contains part, assembly, and component

Figure 12-3.
When placing components, this **Open** dialog box appears.



file options. Selecting **Component Files** displays both part and assembly files located in the specified path. You can also choose to display components created using other CAD files or those saved as STEP or SAT files. The **Use iMate** check box is also available which if selected automatically places a constraint between components with iMates when components are inserted.

NOTE



Imported component files such as STEP or SAT files are not parametrically controlled.

The first component you place in an assembly is automatically grounded and located at the center point. See Figure 12-4. You can automatically place additional ungrounded copies of the base component by picking a location in space to place the component, or pressing [Enter]. If you only need to place one component, or after you place multiple components, right-click and select the **Done** menu option, or press [Esc]. After the initial component is added to the assembly file, all other components, including those that are identical to the first component, are not automatically grounded or constrained, unless they contain iMates.

You may choose to continue inserting parts and subassemblies until all the required components are placed in the assembly file as shown in Figure 12-5. If so, the next step in assembly development is to establish links between individual components using assembly constraints. However, depending on your approach to inserting assembly components and the complexity of the assembly, you may want to insert and constrain one or two components at a time.

If you receive the **Resolve Link** dialog box when you try to insert an existing component, most likely, the component file is not located in the specified project folder, is not available because of a network problem (if the file is contained in a local network), the initial file name was changed, or the server where the file is located is not functioning correctly. To ensure you do not receive the **Resolve Link** dialog box, identify and solve the problem. However, you can pick the **Open** button in an attempt to locate the file elsewhere, or skip the entire process by picking the **Skip** button.



Complete the Exercise on the Student CD.

Figure 12-4.
An initially placed assembly component.

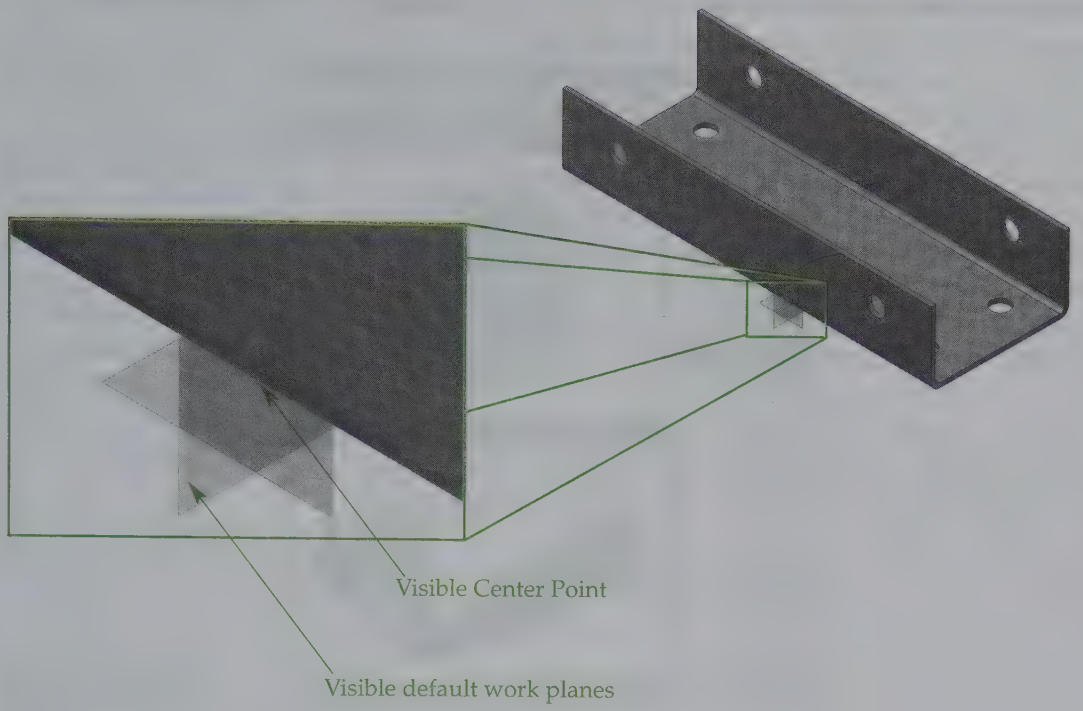


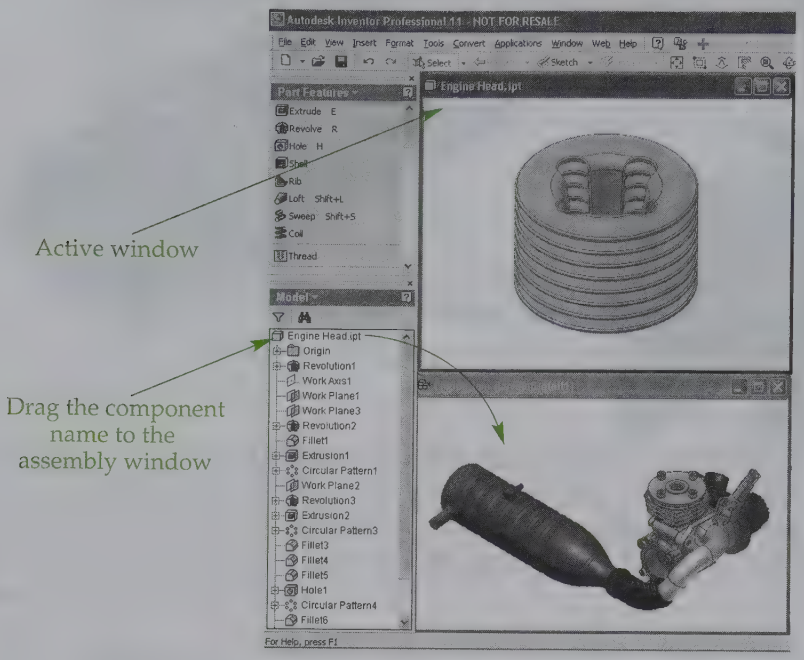
Figure 12-5.
Inserting assembly components.



Alternative Methods for Inserting Assembly Components

Although the typical method for inserting assembly parts and subassemblies involves using the **Place Component** tool, there are alternative techniques for placing assembly components depending on your preference, or the type of assembly you create. One option is to open an assembly file for use in receiving the components, and open one or more components (part or assembly files) that you want to insert into the designated assembly file. Then, arrange the file windows so they are all visible. See Figure 12-6. Next, select the window of a component to make it active. Finally, drag the component icon and name from the browser of the selected component window, and drop the component into the assembly window.

Figure 12-6.
The **Engine Head.ipt** component is inserted into the **Engine.iam** assembly by dragging the component name from the **Browser** into the assembly window.



NOTE

You can only drag and drop one component at a time.

Another technique is to open an assembly file to be used to receive the components. The assembly file can be a default such as Assembly1, a template file, or a saved file with existing components. Then, access Microsoft Windows Explorer, and locate the part or subassembly files you want to insert into the assembly. Next, select all the components you want to insert, drag them to the **Assembly** button on the taskbar, and hold them there until the Inventor assembly window opens, or drag them into the opened Inventor assembly window, if it is displayed. Finally, continue dragging the components into the graphics window, and drop them into place.

Exercise 12-5

Complete the Exercise on the Student CD.

Inserting Shared Content

While you may create most assemblies using parts and subassemblies located on your computer or a local network, you can also access part files (.ipt) and SAT files using the Internet and Intranet. ***Shared content***, also known as ***third-party content***, consists of files available on the Internet, such as bolts from a bolt manufacturer, or components accessible on an Intranet system, such as standard parts that are used for developing assemblies. The shared content capabilities of Inventor are typically used to generate image part libraries accessible to multiple users, to develop assemblies

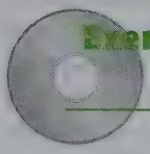
that require components available through the Internet or Intranet, and for companies and vendors located in multiple areas to share design information.

Shared, or third-party content, is also referred to as ***i-drop***, because of the ability to locate .ipt and SAT files through an Internet or Intranet connection and drag and drop the file into an assembly. To i-drop assembly components, first open the assembly file in which you want to place shared content components. Then, using your Web browser, locate the Web site that contains the .ipt or SAT files that you want to insert. Move your cursor over the image of the desired component file and hold down the left mouse button to "fill" the i-drop icon. Now drag the file to the assembly button on the taskbar, and hold it there until the Inventor assembly window opens, or drag the file into the opened Inventor assembly window, if it is displayed. Finally, release the left mouse button in the graphics window, much like placing any other assembly component. Continue dragging components into the graphics window as needed, and drop them into place.

NOTE



Only .ipt and SAT files that are enabled by i-drop can be accessed from a Web site.



Exercise 12-6 Complete the Exercise on the Student CD.

Developing Components In-Place

The other option for creating assemblies involves building components inside the assembly file, also known as creating components ***in-place***. However, usually both methods of inserting existing components and creating components in-place are used to develop complex assemblies. Often, standard base parts or subassemblies are individually built and then inserted into an assembly file, though all components can be generated inside a single assembly file. The advantage of creating components in place is that you can quickly develop an assembly and analyze design intent as you model a product. In addition, when you build a component inside an assembly file, a separate, fully parametric, individual part file (.ipt) or subassembly (.iam) is also generated.

The process for creating parts in an assembly file is identical to creating parts in a part file, except for a few additional processes. All tools available in a part file are found in an assembly file, including the sketch, 3D sketch, features, solids, and sheet metal tools.

If you choose to create assembly components inside an assembly file, the first tool you access is the **Create Component** tool. When you develop a component in-place, the component is created in the assembly file, but also in a separate part or assembly file. As a result, the **Create Component** tool allows you to define the component and component file name, the type of component and component file (part or assembly), the location of the new component file, and the template file used for creating the component.

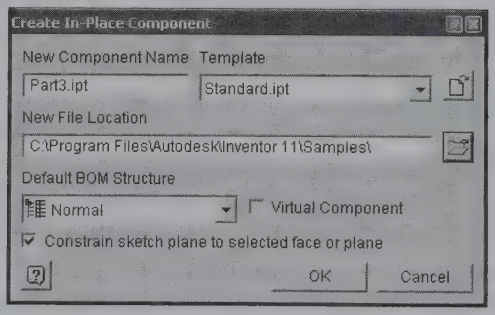
Access the **Create Component** tool by picking the **Create Component** button on the **Assembly Panel** bar; picking the **Create Component** button on the **Assembly Panel** toolbar; selecting the **Create Component...** option from the **Insert** pull-down menu; right-clicking

and selecting the **Create Component** pop-up menu option; or pressing the [N] key on your keyboard. Any of these options opens the **Create In-Place Component** dialog box shown in Figure 12-7. Use the following options in the **Create In-Place Component** dialog box to define the new component:

- **New File Name.** Use this edit box to enter the name of the new component. If you do not enter a different name, a default name is specified, depending on the type of file you create.
- **Template.** Use this drop-down list and the **Browse Templates** button, next to the **Template** drop-down list, to identify the desired template file to use for creating the component file.
- **New File Location.** Use this edit box and the **Browse** button, next to the **New File Location** edit box, to specify the location of the new, separate component file.
- **Default BOM Structure.** This drop-down list allows you to adjust the bill of materials structure status for the individual component file. For more information on the bill of materials refer to the information provided in this chapter.
- **Virtual Component.** Select this check box to create an Inventor virtual component. *Virtual components* are used primarily to define a separate bill of materials item, without creating a model. Virtual components are not models and do not generate separate component files.
- **Constrain sketch plane to selected face or plane.** When this check box is selected, you have the option of establishing a constraint between the sketch plane and the selected existing feature plane. This option is only available when creating a new component in an assembly file that already contains features. Clear this check box if you do not want to specify a constraint between an existing feature plane and the new component.

Once you fully define the options in the **Create In-Place Component** dialog box, pick the **OK** button to begin component development.

Figure 12-7.
The **Create In-Place Component** dialog box.



Creating Parts In-Place

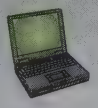
To create a part component, you must specify a part file template in the **Create In-Place Component** dialog box. The process for creating a new part in an assembly varies depending on whether you are creating the first, or base, part or if you are creating an additional part. The process is also slightly different depending on whether you select a specific plane on which to begin the part sketch, or choose the **No New Sketch** radio button from the **Part** tab of the **Options** dialog box.

When creating the base part using a specific sketch plane from the **Part** tab of the **Options** dialog box, first select a location in space, or work feature from the assembly

file **Origin** folder to locate the part. A new sketch is automatically opened on the plane selected in the **Part** tab of the **Options** dialog box, and you enter the 2D sketch environment. When developing the base part using the **No New Sketch** radio button from the **Part** tab of the **Options** dialog box, first select a location in space, or work feature from the assembly file **Origin** folder to define the location of the part. Then, open a sketch on the desired sketch plane, such as one of the default work planes in the **Origin** folder under the part in the browser, to enter the 2D sketch environment. These are essentially the same techniques used to create the first sketch in a part file, except that you must locate the part in the assembly first. As with inserting components, the base component created in-place is grounded.

When adding parts using a specific sketch plane from the **Part** tab of the **Options** dialog box, you can choose to locate the new component sketch in relation to existing components by selecting a feature plane or a work plane. Other options include selecting any of the default work planes available in the assembly **Origin** folder, or choosing not to relate the new sketch to any items by picking a location anywhere in space. Once you pick the location for the new part, a sketch opens and the 2D sketch environment becomes active. When building additional parts using the **No New Sketch** radio button from the **Part** tab of the **Options** dialog box, you must first pick a location for the part by selecting a feature plane or a work plane, any of the default work planes available in the assembly **Origin** folder, or choose not to relate the location of the new part to existing features by picking a location anywhere in space. Once you select the location for the new part, you must then open a sketch on the desired sketch plane, such as one of the default work planes in the **Origin** folder under the part in the browser, to enter the 2D sketch environment.

PROFESSIONAL TIP



If you position a sketch and part by selecting an existing component plane, a flush constraint is automatically placed between the new and existing components. If you do not want to automatically establish a constraint, deselect the **Constrain sketch plane to selected face or plane** check box, previously described, or pick a location in the graphics window not related to any features.

From this point on, creating parts in an assembly file is almost identical to generating parts in a part file, as shown with the browser display in Figure 12-8. Use sketch tools to develop your part sketch. When complete, finish the sketch to exit the sketch environment. Then use feature tools to fully develop your part.

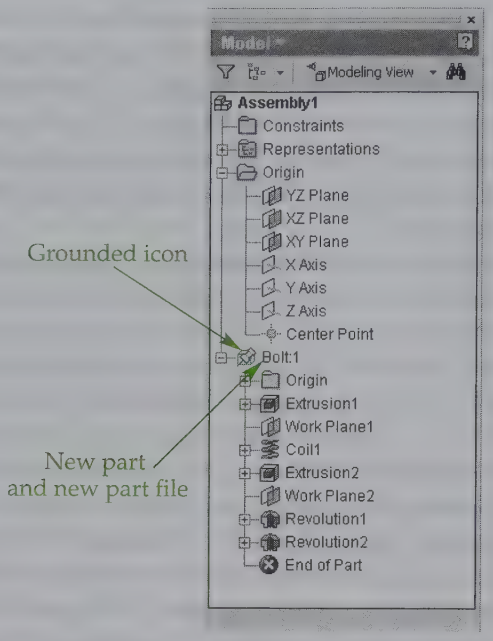
NOTE

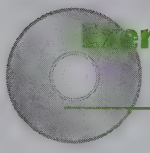


Use the projection techniques described in previous chapters to reference existing features and components when creating a sketch for a new part.

When you create a component in-place, you actually edit the existing assembly file by placing additional content in the form of assembly components. Once you create the part, you must finish the assembly edit and reenter the assembly work environment by right-clicking and select the **Finish Edit** pop-up menu option; selecting the **Return** button on the **Inventor Standard** toolbar; or double-clicking the parent assembly name in the browser. Once you reenter the assembly work environment, you can insert or create additional components, establish constraints between components, or use any of the other tools available.

Figure 12-8.
The browser display in an assembly file named **Assembly1**, and a new part named **Bolt**.





Exercise 12-7 Complete the Exercise on the Student CD.

Creating Subassemblies In-Place

A ***subassembly*** is a group of two or more parts that are linked together to form a portion of a larger assembly. Subassemblies may include standard items like switches or spring assemblies that can be used multiple times in the final assembly. Like the two assembly creation techniques described in this chapter, subassemblies can be formed by inserting existing parts or assemblies, or by building new parts in-place. To create a subassembly component, you must specify an assembly file template in the **Create In-Place Component** dialog box. See **Figure 12-9.** A secondary assembly, or subassembly, is embedded into the assembly file and automatically becomes active.

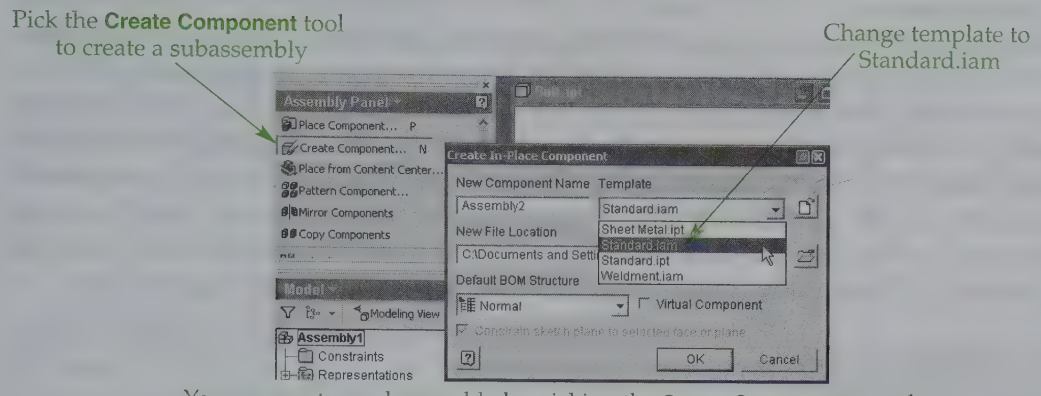
Once you activate a new subassembly using the **Create Component** tool, the process for generating a subassembly is the same as for creating any other type of assembly. You can insert parts or subassemblies using the **Place Component** tool, or develop parts and even additional subassemblies in-place, by accessing the **Create Component** tool again. When you create a component inside a subassembly, or insert a component, the new component is a subassembly item, not the parent assembly file.

NOTE

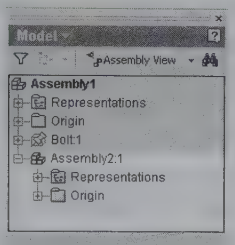


If there are no other components in the assembly file, a new subassembly is automatically placed on the plane selected in the **Part** tab of the **Options** dialog box. However, if you select the **No New Sketch** radio button in the **Part** tab of the **Options** dialog box, you must pick an acceptable plane, such as one of the default work planes in the **Origin** folder, before you enter the subassembly work environment.

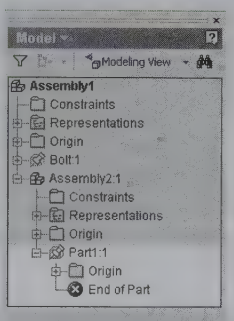
Figure 12-9.
A subassembly becomes active when it is created in an assembly, and a part becomes active when it is created in the subassembly.



You can create a subassembly by picking the **Create Component...** tool



Assembly1 with a new subassembly named Assembly2



Assembly1 with a new subassembly named Assembly2 and a part named Part1 in the subassembly

Continue inserting subassembly components or creating subassembly components in-place using the techniques described. You must ensure the subassembly remains active while you are inserting and creating subassembly components. However, you can apply constraints when either the subassembly is active or the parent assembly file is active. Once you create the subassembly, you must finish the assembly edit and reenter the parent assembly work environment. Do this by right-clicking and selecting the **Finish Edit** pop-up menu option; selecting the **Return** button on the **Inventor Standard** toolbar; or double-clicking the parent assembly name in the browser. Once you reenter the assembly work environment, you can insert or create additional components, establish constraints between components, or use any of the other assembly tools available.

Another option for creating subassemblies is to insert or create components inside the parent assembly file, just like when creating an assembly. Then use one of the following methods to place the components into a separate subassembly file:

- Right-click on the component name in the browser or the component in the graphics window, and select **Demote** from the **Component** cascading submenu.
- Select the component in the graphics window or the component name in the browser and press the [Tab] key on your keyboard.

When you use this method for creating subassemblies, the **Create In-Place Component** dialog box opens, allowing you to specify the subassembly file name and location. When you pick the **OK** button to exit the **Create In-Place Component** dialog box, a subassembly is generated, and the component is automatically placed in the file.

For the opposite application, components can be removed from subassemblies and placed in the parent assembly using one of the following options:

- Right-click on the component name in the browser or the component in the graphics window, and select **Promote** from the **Component** cascading submenu.
- Select the actual component in the graphics window, or the component name in the browser, and press the [Shift] and [Tab] keys on your keyboard.

You can also place components from the parent assembly file into an existing subassembly by dragging the component in the browser from its current location to a location under the desired subassembly **Origin** folder. Similarly, place subassembly components into the parent assembly file by dragging the component from the subassembly and inserting it under the parent assembly **Origin** folder. If you place a component under the **Origin** folder of a subassembly, it is placed in the subassembly.

PROFESSIONAL TIP



Dragging and dropping components into and out of assemblies and subassemblies can become confusing. To confirm the location of components, you may want to collapse all child nodes, then open and close individual subassemblies. Use the **Find** option, available in the **Edit** pull-down menu or by pressing [Ctrl]+[F], to locate components in the browser and in the graphics window.



Exercise 12-8 Complete the Exercise on the Student CD.

Constraining Assemblies

Once you insert or create assembly components, usually the next step in assembly development is to add assembly constraints. Multiple assembly components, including parts and subassemblies, are brought together, or assembled, using constraints. In the same way that constraints define relationships between geometric shapes in the part sketch environment, assembly constraints bring individual and multiple components together by parametrically assigning location specifications to each component in reference to other components. Assembly constraints remove a certain amount of component movement freedom. As a result, constrained components have less or no ability to move, and cannot be located anywhere in space. In addition, any movement freedom still existing can be driven, allowing you to examine various design concepts.

NOTE



You can observe assembly degrees of freedom by dragging components or by selecting the **Degrees of Freedom** option of the **View** pull-down menu.

All components can be constrained to other components in the assembly assuming a preexisting relationship does not affect the constraint. Like all Inventor tools, the types of constraints you use in an assembly are determined by the design. You have

already observed one type of constraint, grounding. Grounded components have no movement freedom, are fixed in position, and cannot be driven. Any component can be grounded, but typically only the first, base component, or components that should not move in reference to other components, are grounded. In addition to grounding, assembly, motion, and transitional constraint are used to fully constrain an assembly. ***Assembly constraints*** establish geometric relationships and positions between one component face, edge, or axis and another component face, edge, or axis. ***Motion constraints*** define the desired movement between one component and another, using a specified ratio and direction. ***Transitional constraints*** identify relationships between the transitioning path of a fixed component and a component moving along the path.

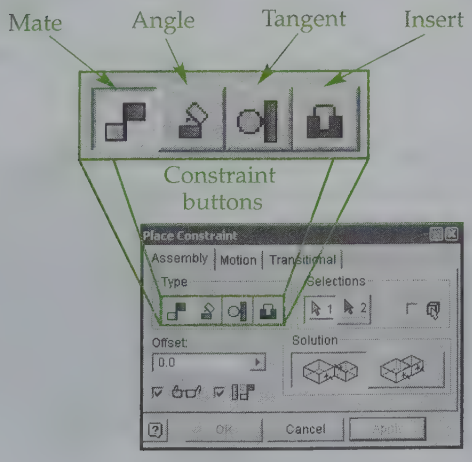
Placing Constraints

One of the most common techniques for assembling components is to use the **Place Constraint** tool, which uses the typical Inventor dialog box approach. The **Place Constraint** tool allows you to establish assembly, motion, and transitional constraints between components. In many situations, you can access the **Place Constraint** tool once, and fully constrain an entire assembly. However, you can use the tool any time during assembly development. Access the **Place Constraint** tool by picking the **Place Constraint** button on the **Assembly Panel** bar; picking the **Place Constraint** button on the **Assembly Panel** toolbar; selecting the **Constraint...** option of the **Insert** pull-down menu; or pressing [C]. This opens the **Place Constraint** dialog box. See Figure 12-10. The first step in constraining an assembly using the **Place Constraint** dialog box is to select the tab that corresponds to the type of constraint you want to create. Define assembly, motion, or transitional constraints by picking the corresponding tab.

Placing assembly constraints

The constraint options available in the **Assembly** tab of the **Place Constraint** dialog box represent the most commonly used constraints for most assembly applications. Almost all assemblies contain at least one assembly constraint, but usually several. In addition, typically before you add motion or transitional constraints, you place assembly constraints. Choose one of the following assembly constraint-type buttons from the **Type** area of the **Assembly** tab, and use the information provided to create the required constraint:

Figure 12-10.
The **Place Constraint** dialog box, **Assembly** tab, **Mate** constraint button selected.



- **Mate.** Select this button to establish a mate or flush constraint. *Mate* and *flush constraints* are common in many assemblies and are used to define a mate or flush relationship between the face, plane, axis, edge, or point of one component, and the face, plane, axis, edge, or point of another component. See Figure 12-11. When you pick the **Mate** button, the **Assembly** tab of the **Place Constraint** dialog box, shown in Figure 12-10, displays mate constraint options. Though you may develop your own approach to applying constraints, typically the first step is to identify the mate constraint solution, by selecting either the **Mate** or **Flush** button of the **Solution** area, though you can change the solution any time during constraint development. A mate solution places two faces along the same plane facing in opposite directions, two axes collinear to each other, two edges collinear to each other, or two points matched together. See Figure 12-12. In contrast, a flush solution positions two faces along the same plane, facing the same direction. See Figure 12-13.

  Once you determine the mate solution you want to use and pick the appropriate solution button, you can begin selecting component features to constrain. If you are working on a complex or detailed assembly, you may find it helpful to choose the **Pick Part First** check box. When selected, the **Pick Part First** check box allows you to pick a part first, isolating the part from other assembly components and making it easier to choose specific geometry on the selected part. Once you pick the part or if the **Pick Part First** check box is not active, you can choose the required component object, including a face, edge, axis, or point. See Figure 12-14. By default, the **First Selection** button is initially active, allowing you to select the first component element. Once you choose the first object on one component, the **Second Selection** button becomes active, allowing you to select an object on the other component. Again, if the **Pick Part First** check box is selected, you must pick the second component before you can make the second selection.

  The final step in applying a mate constraint is to specify an offset between the selected components by entering or selecting a value using the **Offset** edit box. For many applications, an offset of 0 is used to specify a contacting fit, with no offset between components. Another option for defining offset and orientation is to select the **Predict Offset and Orientation** check box before you define a solution and pick components. This method allows Inventor to predict

Figure 12-11.
This is an example of three mate constraints between two parts.

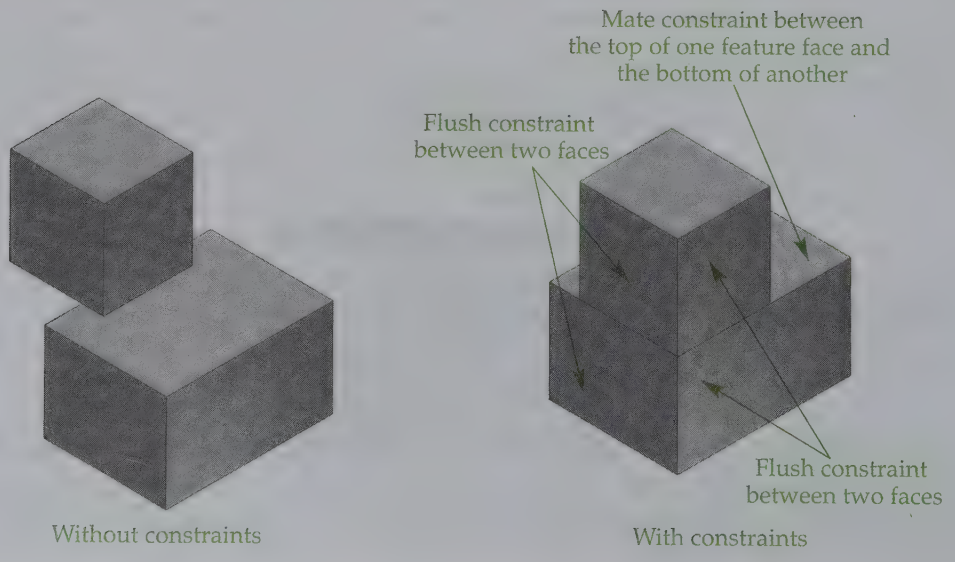
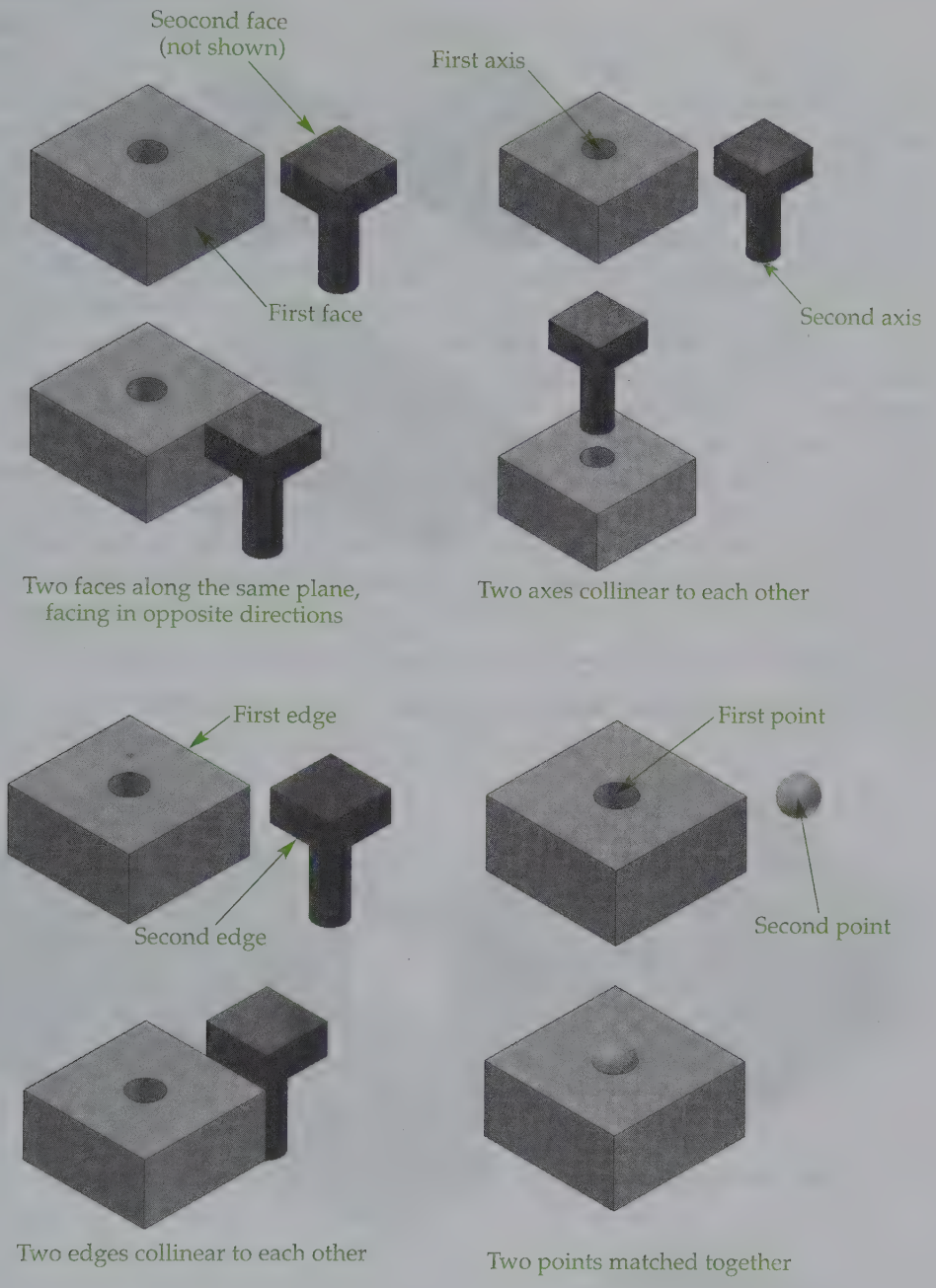


Figure 12-12.
The **Mate** constraint can be used with planar faces, axes, edges, and points.



how components should be constrained based on the components current location in space and your selections. If the selected component arrows are pointing in the same direction, a flush constraint is predicted and the **Flush** button is automatically selected. In addition, assuming no offset is defined, an offset value is placed in the **Offset** edit box based on the current offset of the two selected components. If the selected component arrows are pointing in opposite directions, a mate constraint is predicted and the **Mate** button is automatically selected. Again, assuming no offset is defined, an offset value is placed in the **Offset** edit box based on the current offset of the two selected components. To preview the effects of the constraint, select the **Show Preview** check box. Then, if the constraint looks acceptable, pick the **Apply** button to generate the constraint. When you pick the **Apply** button, the **Place Constraint**

Figure 12-13.
The **Flush** constraint places the selected faces in the same plane and facing the same direction.

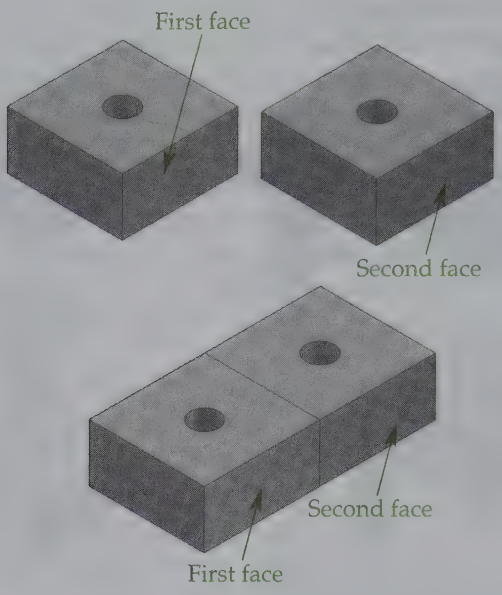
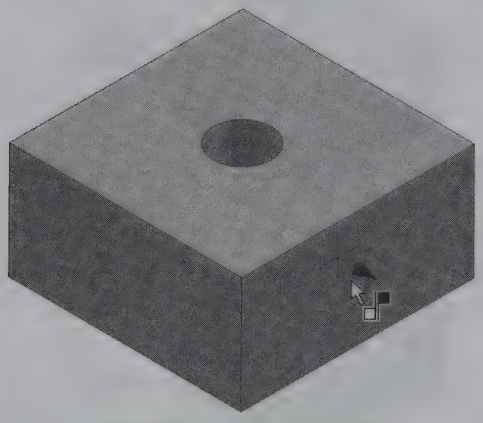
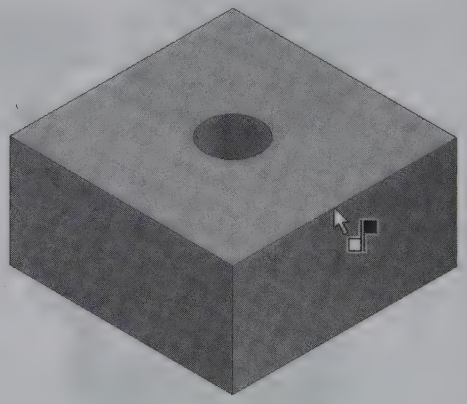
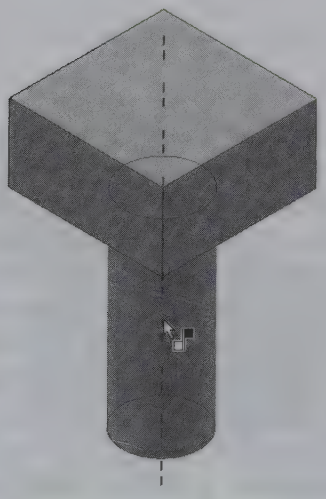


Figure 12-14.
Select one of these component elements after selecting the component.


Selecting a component face


Selecting a component edge


Selecting a component axis


Selecting a component point

tool is still active, allowing you to continue placing constraints. Pick the **OK** button when finished placing constraints, or to generate a constraint and exit the **Place Constraint** tool.

- **Angle.** This constraint allows you to specify an angle between a face, axis, or edge of one component and a face, axis, or edge of another component. See Figure 12-15. When you pick the **Angle** button, the **Assembly** tab of the **Place Constraint** dialog box and angle constraint options are displayed. See Figure 12-16. Typically, the first step in defining an angle constraint is to pick component geometry to constrain. If you are working on a complex or detailed assembly, you may find it helpful to pick the **Pick Part First** check box, which isolates the selected component from other assembly components and makes choosing specific features on the specified part easier.
  Once you select the part or if the **Pick Part First** check box is not active, you can select component elements including faces, edges, and axes. See Figure 12-17. By default, the **First Selection** button is initially active allowing you to select the first component object. Once you choose the first element on one component, the **Second Selection** button becomes active allowing you to select the second object on the other component. Again, if the **Pick Part First** check box is selected, you must pick the second component before you can make the second feature selection. Any combination of faces, edges, or axes can be selected depending on design intent. Figure 12-17 shows the effects of creating a 0° angle when you

Figure 12-15.
An example of a 45° angle constraint between two selected faces.

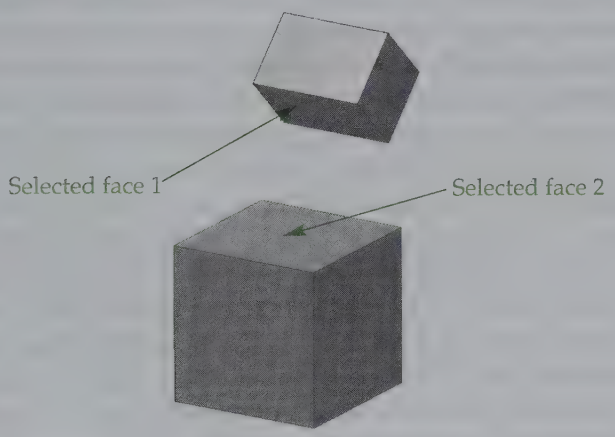


Figure 12-16.
This is the **Place Constraint** dialog box, **Assembly** tab.

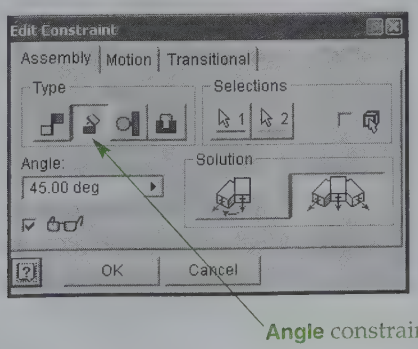
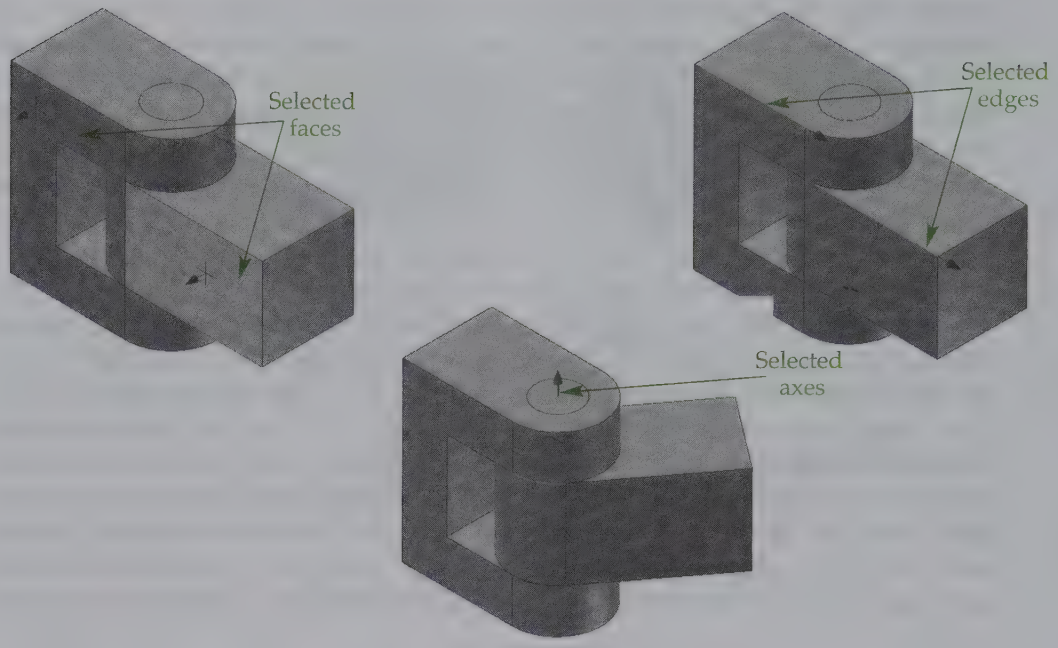


Figure 12-17.
These are examples of selecting various component elements.



select the two perpendicular objects shown, while Figure 12-18 displays the effects of creating a 0° angle when alternative object combinations are selected. To preview the effects of the constraint, select the **Show Preview** check box. If the constraint does not look acceptable, you may need to revise your selections, or pick the **Directed Angle** button instead of the default **Undirected Angle** button from the **Solution** area before you choose components. The selected solution button may also help you define angle constraint drive characteristics.

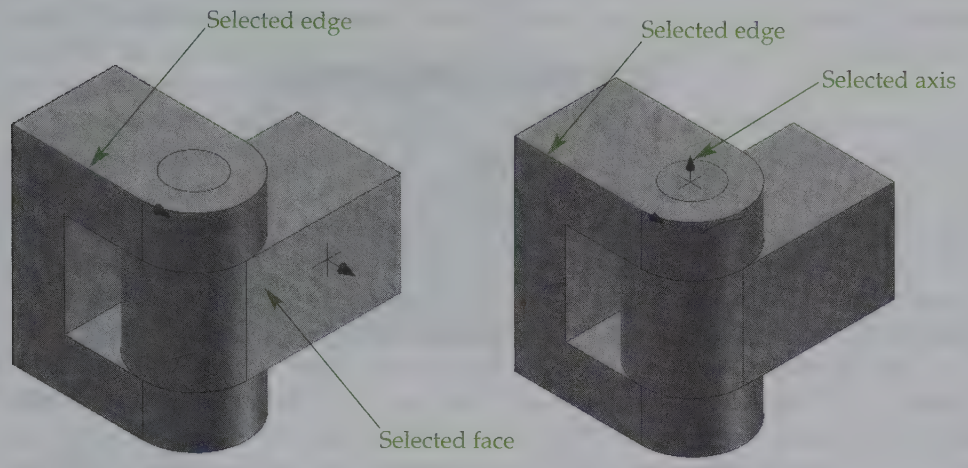
The final step in applying an angle constraint is to specify an angle between the selected components. For some applications, an angle of 0° is specified in the **Angle** edit box, such as for driving purposes. However, often a specific angle is specified to incorporate design intent. Another option for defining angle and orientation is to select the **Predict Offset and Orientation** check box before you define a solution and pick components. This method allows Inventor to predict how components should be constrained based on the component's current location in space and your selections. For example, assuming no angle is defined, if the selected component arrows are pointing in opposite directions, an angle of 180° is automatically specified in the **Angle** edit box. If the constraint looks acceptable, pick the **Apply** button to generate the constraint. When you pick the **Apply** button, the **Place Constraint** tool is still active, allowing you to continue placing constraints. Pick the **OK** button when finished placing constraints or to generate a constraint and exit the **Place Constraint** tool.

NOTE



As with all assembly component constraint selection options, there are multiple ways to identify component objects for constraining. You should determine which component and button selections are needed depending on the application and design intent.

Figure 12-18.
Selecting alternative component objects displays these previews.



- **Tangent**. Use this constraint to define a relationship between one component face or edge, and the tangency of a curved component feature face, such as a cone, cylinder, sphere, or other round face. You can also establish a constraint between two curved component feature faces. See Figure 12-19. When you pick the **Tangent** button, the **Assembly** tab of the **Place Constraint** dialog box displays tangent constraint options. See Figure 12-20. Typically, the first step in establishing a tangent constraint is to identify the tangent constraint solution by selecting the **Inside** or **Outside** button in the **Solution** area, although you can change the solution any time during constraint development. An inside solution places the component you select first inside the component you select second. In contrast, an outside solution positions the first component you select outside of the second component selected. See Figure 12-21. Once you determine the

Figure 12-19.
The **Tangent** constraint can be used between two curved surfaces and between a curved and a planar surface.

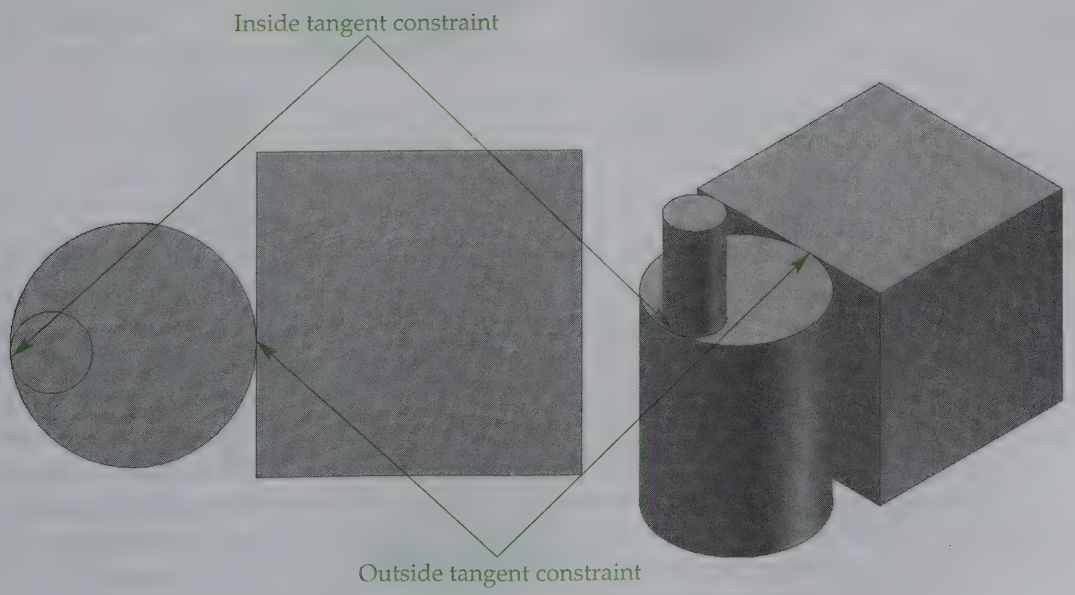


Figure 12-20.
The **Place Constraint** dialog box, **Assembly** tab.

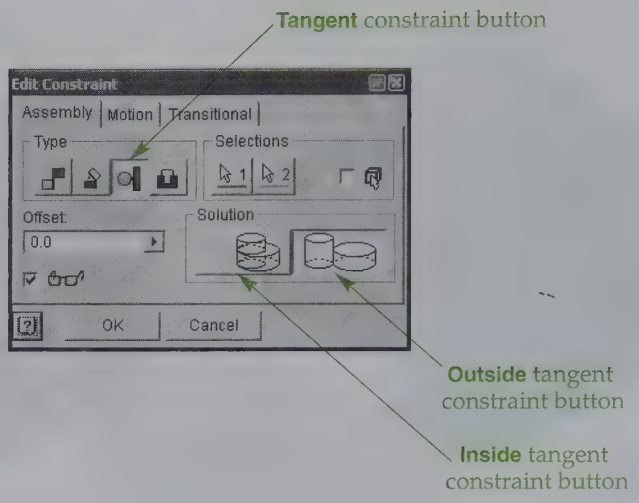
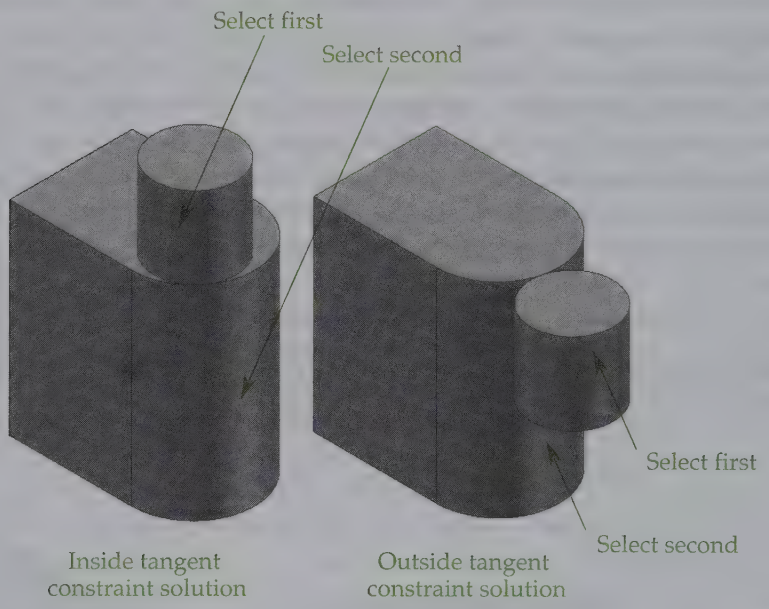


Figure 12-21.
The first object selected will be inside or outside the second, depending on which solution button was activated.



tangent solution you want to use and pick the appropriate solution button, you can begin selecting faces or edges to constrain components. It you are working on a complex or detailed assembly, you may find it helpful to pick the **Pick Part First** check box, which isolates the selected part from other assembly components and makes choosing specific features on the specified part easier. Once you select the part, or if the **Pick Part First** check box is not active, you can select the required component objects. By default, the **First Selection** button is initially active, allowing you to select the first component element. You can choose an edge, planar face, or a curved face for the first selection. If the first selection is an edge or planar face, the second selection must be a tangent face. Conversely, if the second selection is an edge or planar face, the first selection must be a tangent face.

See Figure 12-22. Once you choose the first object on one component, the **Second Selection** button becomes active, allowing you to select the second object on the other component. Again, if the **Pick Part First** check box is selected, you must pick the second component before you can make the second component element selection. As previously described, the first component you select moves to constrain the second selected component, which does not move.

The final step in applying a tangent constraint is to specify an offset between the selected components by entering or selecting a value in the **Offset** edit box. For many applications, an offset of 0 is used to specify a contacting fit. To preview the effects of the constraint, select the **Show Preview** check box. Then, if the constraint looks acceptable, pick the **Apply** button to generate the constraint. When you pick the **Apply** button, the **Place Constraint** tool is still active, allowing you to continue placing constraints. Pick the **OK** button when finished placing constraints, or to generate a constraint and exit the **Place Constraint** tool.

- **Insert.** Use this constraint to establish a relationship between two cylindrical objects, such as a hole and a shaft. An insert constraint is similar to defining both axis-to-axis and edge-to-edge mate constraints. See Figure 12-23. Using

Figure 12-22.
These simple assemblies show examples of outside tangent constraints.

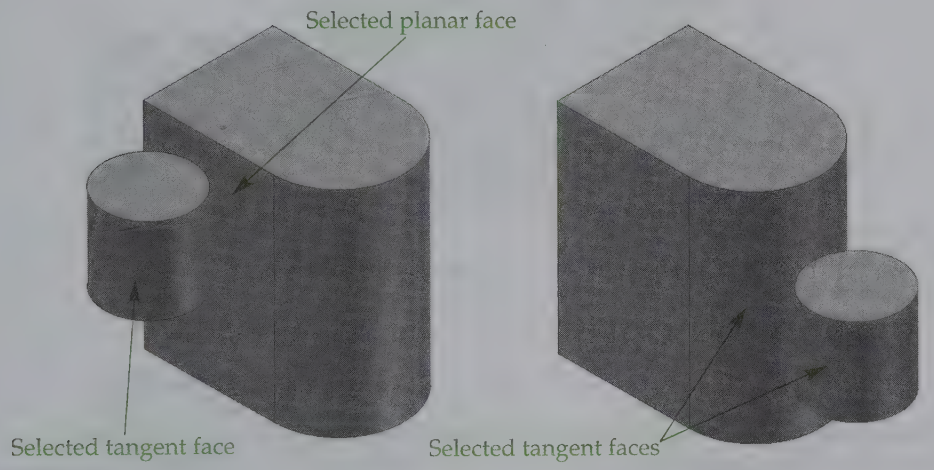
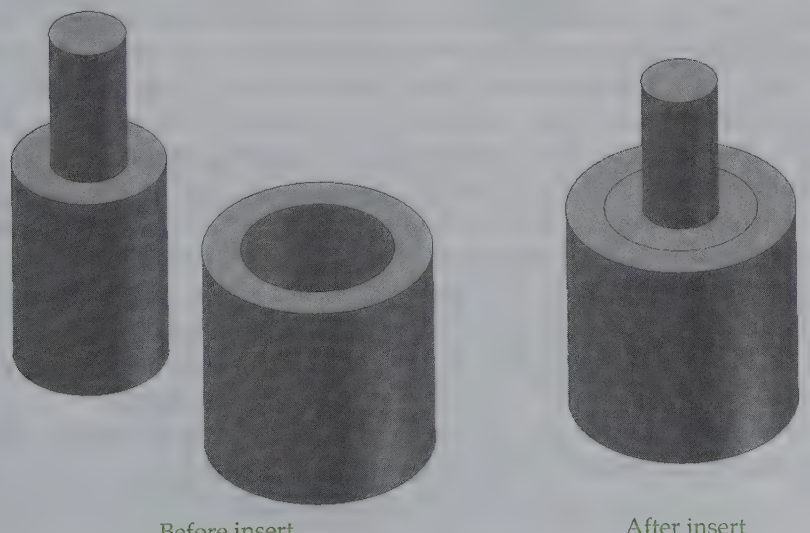
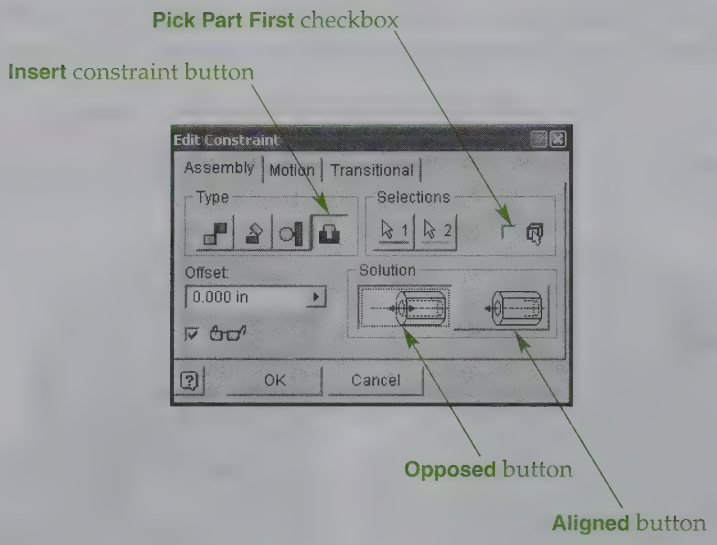


Figure 12-23.
The **Insert** constraint centers a shaft inside a hole.



an insert constraint is an easy way to link two components that may require both an axis-to-axis and edge-to-edge mate constraint, such as the constraints typically found between a screw and a hole. In this example, a constraint is established between the axis of the screw and the axis of the hole, and the edge of the hole and the inside edge of the screw head. When you pick the **Insert** button, the **Assembly** tab of the **Place Constraint** dialog box displays the insert constraint options. See Figure 12-24. Typically the first step in defining an insert constraint is to pick component geometry. If you are working on a complex or detailed assembly you may find it helpful to pick the **Pick Part First** check box, which isolates the selected part from other assembly components and makes choosing specific features on the specified part easier.

Figure 12-24.
The **Place Constraint** dialog box, **Assembly** tab.



Once you select the part, or if the **Pick Part First** check box is not active, you can select components to constrain. See Figure 12-25. By default, the **First Selection** button is initially active, allowing you to select the first component. Once you choose the first component, the **Second Selection** button becomes active, allowing you to select the second component. Again, if the **Pick Part First** check box is selected, you must pick the second component before you can make the second selection. An insert selection picks both the axis and the specified edge of a component. You must select the corresponding edges on both components to ensure the proper mate constraint is generated between components. To preview the effects of the constraint, select the **Show Preview** check box. If the constraint does not look acceptable, you may need to flip the direction of the inserted component by picking the **Aligned** button in the **Solution** area. Many insert constraints use an **Opposed** solution, while others require an aligned solution. See Figure 12-26. The next step in applying an insert constraint is to specify an offset between the selected components by entering or selecting a value using the **Offset** edit box. For many applications, an offset of 0 is used to specify a contacting fit, with no offset between components.

Finally, to generate the constraint, pick the **Apply** button. When you pick the **Apply** button, the **Place Constraint** tool is still active, allowing you to continue placing

Figure 12-25.
Selecting components to apply an insert constraint.

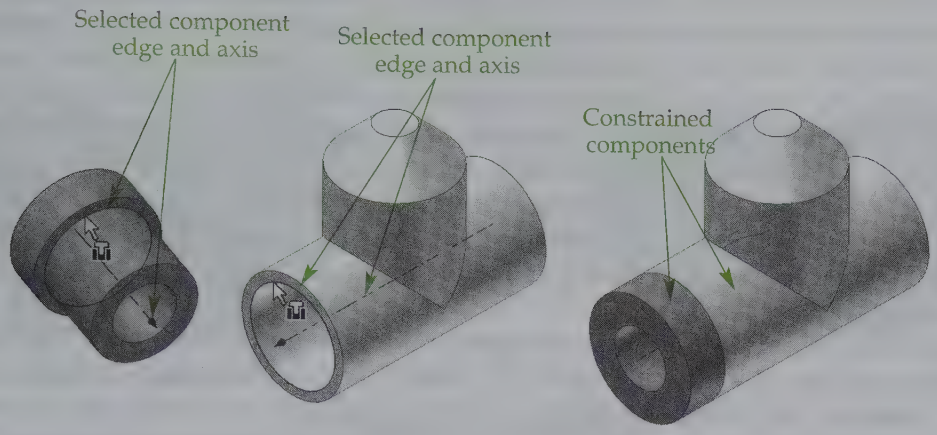
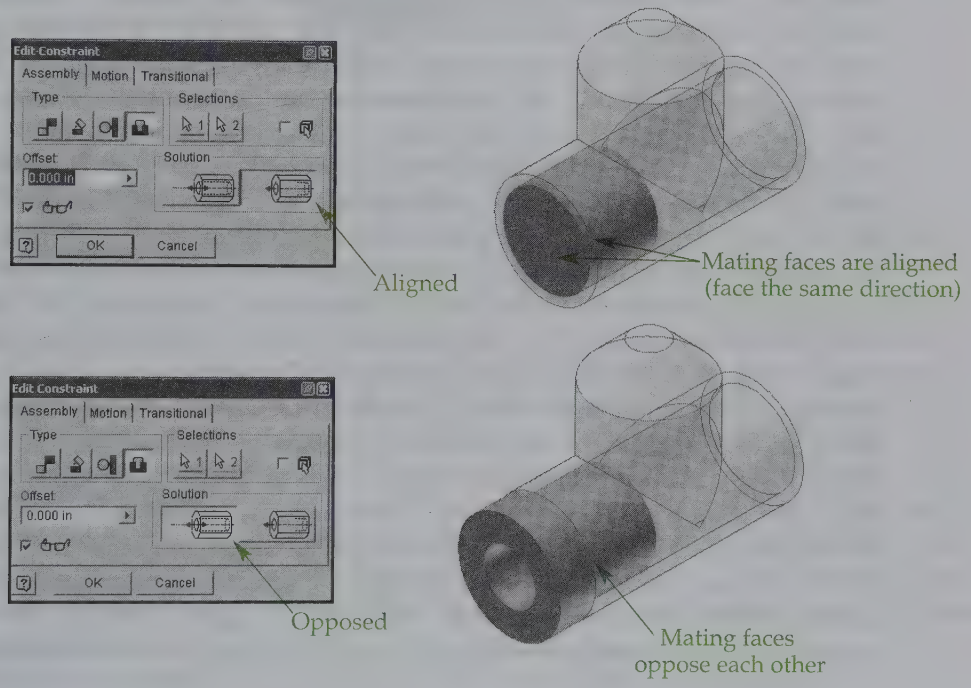
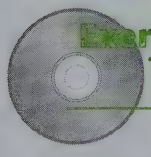


Figure 12-26.
Selecting an **Aligned** or **Opposed** solution determines the direction of the insert.



constraints. Pick the **OK** button when finished placing constraints, or to generate a constraint and exit the **Place Constraint** tool.



Exercise 12-9 Complete the Exercise on the Student CD.

Adding motion constraints

The **Motion** tab of the **Place Constraint** dialog box, shown in Figure 12-27, contains options that allow you to define the movement between one component and another,

using a specified ratio and direction. Only components that rotate and move together, such as gears, pulleys, racks and pinions, bearings, and other rotating components, can contain motion constraints. Motion constraints do not define the location or position of components and do not fix components in space. The purpose of motion constraints is to identify how movable components should move in reference to other movable components. For example, a motion constraint can identify whether two gears move in the same direction or opposite directions, and specify the ratio of rotation, such as 1:1. Consequently, you must use assembly constraints to establish all other geometric constraints, such as a flush relationship between the tops of two gears. Assembly constraints are usually added before motion constraints. In addition, you may need to suppress or drive assembly constraints that do not allow for movement in order to observe a motion constraint.

Choose one of the following motion constraint type buttons from the **Type** area of the **Motion** tab, and use the information provided to create the required constraint:

- **Rotation.** Select this button to establish a direction of rotation and a rotation ratio for two revolving components, such as gears, pulleys, and wheels. When the **Rotation** button is selected, the **Motion** tab of the **Place Constraint** dialog box contains options that allow you to define a rotation motion constraint. See Figure 12-27. As with other constraining options, if you are working on a complex or detailed assembly, you may find it helpful to first pick the **Pick Part First** check box to isolate the selected part from other assembly components. Once you select the part, or if the **Pick Part First** check box is not active, you can select the required component objects including edges, axes, planar faces, and tangent faces. By default, the **First Selection** button is initially active, allowing you to select the first component element. Once you choose the first object on one component, the **Second Selection** button becomes active, allowing you to select the second object on the other component. Again, if the **Pick Part First** check box is selected, you must pick the second component before you can make the second component element selection. Selecting certain component elements is not as critical when applying motion constraints, because motion constraints do not define the position of components in reference to other components. As a result, Figure 12-28 shows a few examples of selection options that create the same constraint between two gears. However, the pattern of selection does define the specified ratio. So, if you define a 3:1 ratio, the first component you select should be the component that revolves once for every three times the second component rotates. The next step in defining a rotation motion constraint is to specify the rotation solution. By default, the **Forward** solution button is selected, which creates a forward motion between the selected components. A forward motion allows one component to rotate in the same direction as the other component. The arrows shown in Figure 12-28 identify the rotation direction. To

Figure 12-27.
The **Place Constraint** dialog box, **Motion** tab.

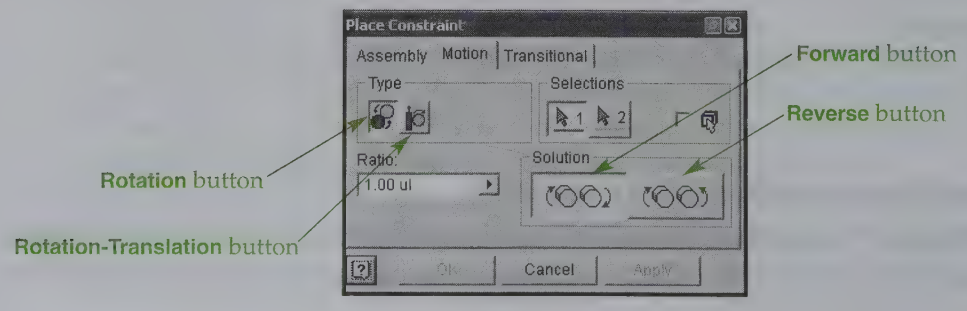
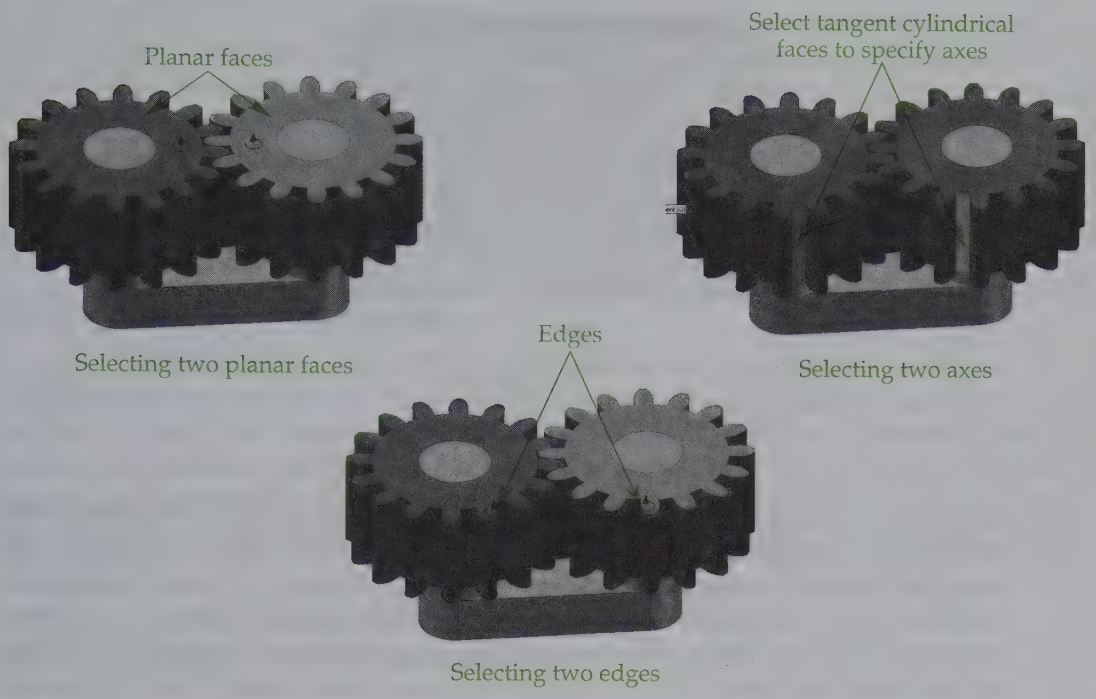


Figure 12-28.
All of these selection types will create the same rotational constraint between the gears.



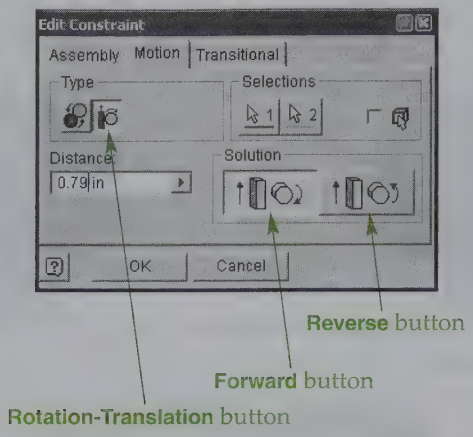
reverse the rotation, pick the **Reverse** solution button. A reverse motion allows one component to rotate in the opposite direction of the other component.

The final step in applying a rotation motion constraint is to specify a rotation ratio between the selected components by entering or selecting a value in the **Ratio** edit box. The specified ratio defines how much the second selected component rotates in reference to the first component selected. For example, if you apply a rotation constraint between the same size gears, as shown in Figure 12-28, a ratio of 1:1, or 1 in the **Ratio** edit box, is appropriate. If the first selected gear, wheel, or pulley is half the size of the second component and rotates twice for every single rotation of the second component, a ratio of 1:2, or .5, in the **Ratio** edit box is used. To generate the constraint, pick the **Apply** button. Again, when you pick the **Apply** button, the **Place Constraint** tool is still active, allowing you to continue placing constraints. Pick the **OK** button when finished placing constraints, or to generate a constraint and exit the **Place Constraint** tool.

- **Rotation-Translation.** Select this button to establish a direction of rotation and a rotation distance between a rotating component and a translating component, such as a rack and pinion, or a wheel and part face. When the **Rotation-Translation** button is selected, the **Motion** tab of the **Place Constraint** dialog box contains options that allow you to define a rotation-translation motion constraint. See Figure 12-29. As with other constraint options, if you are working on a complex or detailed assembly, you may find it helpful to first pick the **Pick Part First** check box to isolate the selected part from other assembly components.

  Once you select the part, or if the **Pick Part First** check box is not active, you can select the required component objects including edges, axes, planar faces, and tangent faces. By default, the **First Selection** button is initially active, allowing you to select the first component element. Once you choose the first object on one component, the **Second Selection** button becomes active, allowing you to

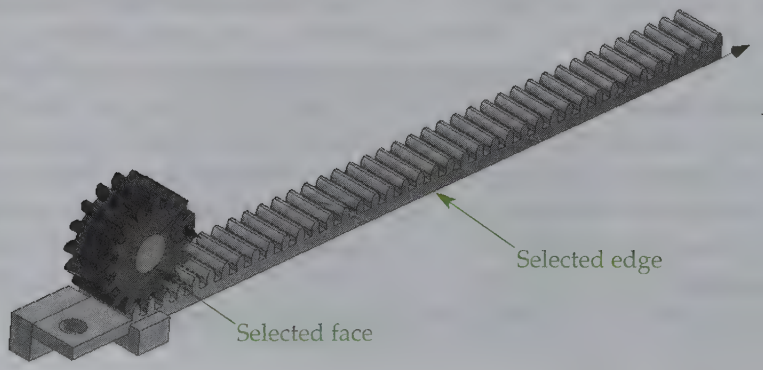
Figure 12-29.
The **Place Constraint** dialog box, **Motion** tab, **Rotation-Translation** constraint button selected.



select the second object on the other component. Again, if the **Pick Part First** check box is selected, you must pick the second component before you can make the second component element selection. Selecting certain component elements is not as critical when applying motion constraints, because motion constraints do not define the position of components in reference to other components. As a result, Figure 12-30 shows two examples of selection options that create the same constraint between a rack-and-pinion assembly. However, the pattern of selection does define the specified distance. So, if you define a distance of 3"(73.5 mm), the first component you select revolves 360° for every 3" the second component moves. The next step in defining a rotation-translation motion constraint is to specify the rotation-translation solution. By default, the **Forward** solution button is selected, which creates a forward motion between the selected components. A forward motion allows the rotating component to rotate in the same direction as the sliding component. The arrows shown in Figure 12-30 identify the rotation direction. To change the component motion to a reverse rotation, pick the **Reverse** solution button. A reverse motion allows the rotating component to rotate in the opposite direction of the sliding component.

The final step in applying a rotation-translation motion constraint is to specify a rotation-translation distance between the selected components by entering

Figure 12-30.
Selecting a planar face and an edge to add rotation-translation constraint.



or selecting a value in the **Distance** edit box. The specified distance defines how far the sliding component, second selection, moves in reference to the rotating component, first selection. For example, if you apply a rotation-translation constraint of 3", as previously mentioned, for every full rotation of 360°, the first component revolves, the second component moves 3". In the same respect, a distance of 0 does not allow the second component to move in reference to the revolution of the first component. To generate the constraint, pick the **Apply** button. When you select the **Apply** button, the **Place Constraint** tool is still active, allowing you to continue placing constraints. Pick the **OK** button when finished placing constraints, or to generate a constraint and exit the **Place Constraint** tool.

Adding transitional constraints

The **Transitional** tab of the **Place Constraint** dialog box, shown in Figure 12-31, contains options that allow you to define the desired movement between a moving part and a path. The clearest example of a transitional constraint is between a moving cylinder and a stationary route, such as a roller and a cam. See Figure 12-32. By selecting two planar or tangent edges, a transitional constraint links faces together, applying constant contact between two components. Transitional constraints help define

Figure 12-31.
The **Place Constraint** dialog box, **Transitional** tab.

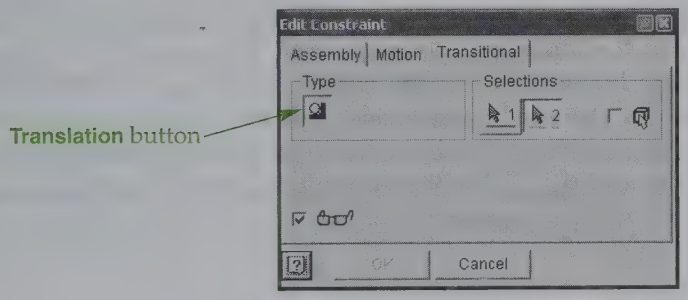
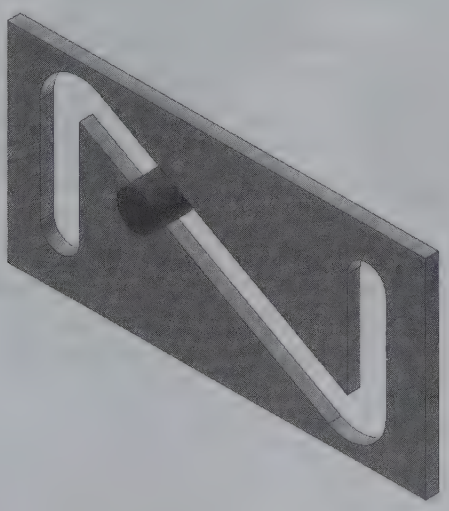


Figure 12-32.
An example of a transitional constraint application, where the roller moves along the faces of the cam.



the location or position of components by adding a type of mate constraint between faces, but do not fix components in space. The purpose of a transitional constraint is to identify how a movable component should follow a certain path. Consequently, you must use assembly constraints to establish all other types of constraints, such as the flush relationship between the top of the roller and the top of the cam in Figure 12-32. Assembly constraints are usually added before transitional constraints and may need to be suppressed or driven in order to show transitional movement.

To establish a transitional constraint, as with other constraining options, you may find it helpful to first pick the **Pick Part First** check box. Use this option if you are working on a complex or detailed assembly in order to isolate the selected part from other assembly components. Once you select the part, or if the **Pick Part First** check box is not active, you can choose the required component objects, including planar or tangent faces. By default, the **First Selection** button is initially active, allowing you to select the first component element. Once you choose the first object on one component, the **Second Selection** button becomes active, allowing you to select the second object on the other component. Again, if the **Pick Part First** check box is selected, you must pick the second component before you can make the second component face selection. The first component you select moves to constrain the second component selected, which does not move. To preview the effects of the constraint, select the **Show Preview** check box. Then, if the constraint looks acceptable, pick the **Apply** button to generate the constraint. When you pick the **Apply** button, the **Place Constraint** tool is still active, allowing you to continue placing constraints. Pick the **OK** button when finished placing constraints, or to generate a constraint and exit the **Place Constraint** tool.

PROFESSIONAL TIP



Help locate geometry to constrain using the **Select Other** tool and wheel mouse selection techniques described in previous chapters. You can cycle through possible edges, axes, planes, and other elements for constraint purposes.



Exercise 12-10 Complete the Exercise on the Student CD.

Constraining by Dragging Components

Components can be constrained using a technique known as ***drag-mate*** or ***alt-drag***. This method establishes various assembly constraints, including mate, flush, tangent, and insert constraints, by dragging one component to another component. Like other constraining options, you cannot drag-mate grounded components or those that are already fully constrained. To move an unconstrained component, you can hold down the left mouse button and drag the component to the desired location. Establishing mate constraints between components is a similar process. However, before you pick the component, you must first hold down the [Alt] key on your keyboard. Then, while still pressing the [Alt] key, pick the component face, edge, or point you want to move into place, hold down the left mouse button, and drag the component to the other

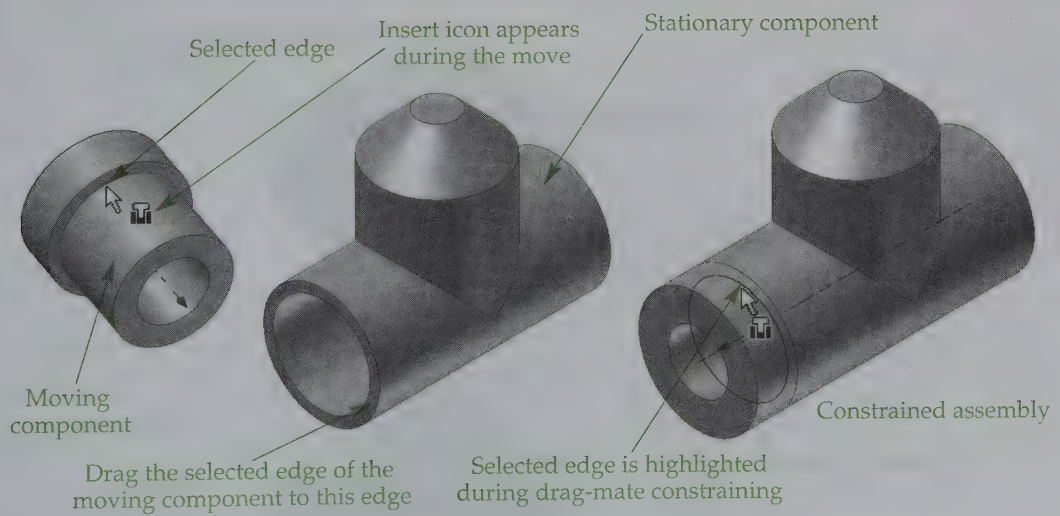
component face, edge, or point to which you want to constrain. As you pass the selected component over and around other components, the constraint options are previewed, along with the possible constraint icon.

Often the automatic constraint option available when using the drag-mate constraint technique is not correct. If the incorrect drag-mate constraint is shown, you may be able to establish the correct constraint by moving the component slightly, or you can toggle between different types of constraints using hot keys. To use hot keys, while you are holding down the left mouse button, release the [Alt] key on your keyboard, and select one of the following keys to activate the specified constraint and solution:

- [A] or [2]. Select either key to change the current constraint to an angle constraint. Then, press the spacebar to change the angle direction, if required.
- [I] or [4]. Select either key to change the current constraint to an insert constraint. Then, if needed, press the spacebar to change the insert direction.
- [M] or [1]. Select either key to change the current constraint to a mate constraint. Then, if required, press the spacebar to change a flush solution.
- [R] or [5]. Select either key to change the current constraint to a rotation motion constraint. Then, if needed, press the spacebar to change the rotation direction.
- [S] or [6]. Select either key to change the current constraint to a translation motion constraint. Then, press the spacebar to change the slide direction, if needed.
- [T] or [3]. Select either key to change the current constraint to a tangent constraint. Press the spacebar to change to an inside or outside tangent constraint.
- [X] or [8]. Select either key to change the current constraint to a transitional constraint.

Once you are satisfied with the intended constraint, release the left mouse button to generate the constraint. See **Figure 12-33.** To ensure the constraint is correct, look in the browser to observe the drag-mate constraint.

Figure 12-33.
Hold down the [Alt] key and click-and-drag the face, edge, or point of the moving component to the corresponding location on the stationary component.



PROFESSIONAL TIP



Once you begin the drag-mate constraint process, you can help locate geometry to constrain, using the **Select Other** tool and wheel mouse selection techniques described in previous chapters. You can cycle through possible edges, axes, planes, and other elements for constraint purposes.

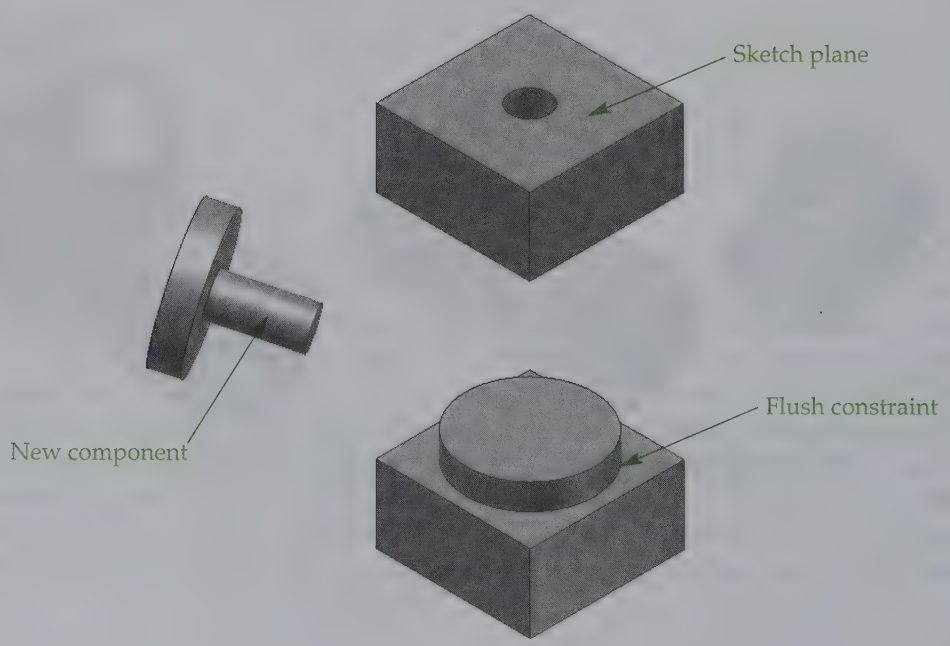
Constraining Components In-Place

The first step in creating parts and subassemblies in-place is to identify a sketch location. The sketch can be placed anywhere in space or can be added to an existing component face or plane. When you sketch on an existing component face, a flush constraint is automatically created between the sketch plane of the new component and the face of the existing component. See Figure 12-34. In addition, when you create a component in-place using existing component geometry, the new component automatically becomes adaptive, as indicated in the browser. The adaptive nature of the components in Figure 12-34, for example, allows the shaft to become constrained to the hole. If you remove the adaptivity between the shaft and the hole, the only constraint still present is the flush constraint. As a result, you must specify a mate constraint between the shaft and hole axes in order to keep the shaft positioned in the hole. Adapting parts and assemblies is described later in this text.



Exercise 12-11 Complete the Exercise on the Student CD.

Figure 12-34.
An example of a constraint automatically placed between the sketch plane of a new component and the face of an existing component.

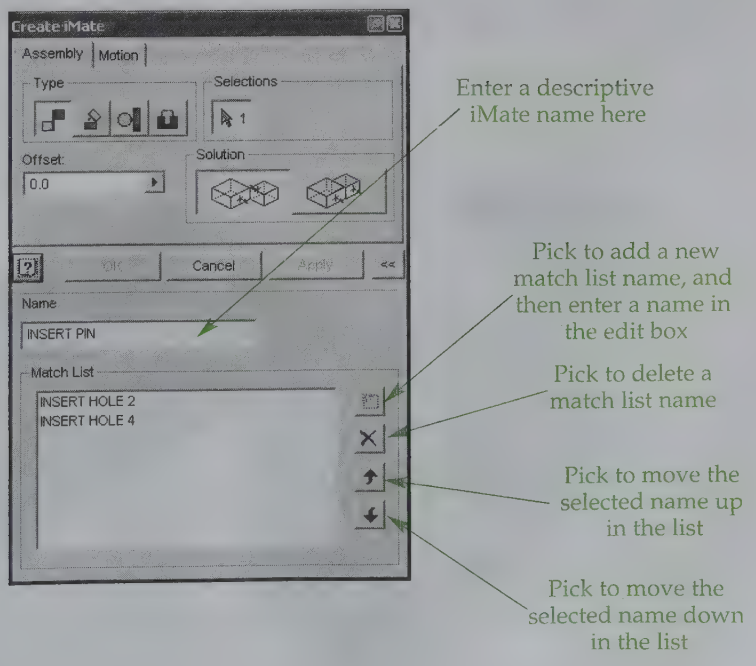


Creating iMates

Most constraints are applied to two components at the same time while in an assembly file. For example, using the **Place Constraint** tool, you can establish an insert constraint between a bolt and a part with a threaded hole. Another option for identifying constraints is to use iMates. ***iMates*** allow you to define one or more constraints on the component that are then used in the assembly. You can think of an individual iMate as half of a constraint that is added to a component when it is created or edited. The two constraint halves, or iMates, can then be linked together, forming an assembly constraint. Using the previous example, you can place an insert iMate when you create or edit the bolt and an insert iMate when you create or edit the part with the threaded hole. Then, when both the bolt and the part with the threaded hole are placed in an assembly file, they can be constrained together using the predefined iMates. Probably the most difficult aspect of working with iMates involves planning and identifying iMate placement. If you do not use iMates that correspond to other iMates or do not place iMates in the correct locations in reference to other components, incorrect constraining occurs.

The first step to applying iMate relationships is to create iMates for each of the components you want to constrain together. To create iMates, you must open a part or subassembly file or edit an existing component in the assembly file by right-clicking on the component name in the browser or the actual component in the graphics window and selecting the **Edit** menu option. Then, access the **Create iMate** tool by right-clicking inside the graphics window and selecting the **Create iMate** pop-up menu option; right-clicking on an existing **iMate** folder in the browser and selecting the **Create iMate** pop-up menu option; selecting the **Create iMate** option of the **Tools** pull-down menu; picking the **Create iMate** button on the **Part Features** panel bar; picking the **Create iMate** button on the **Part Features** toolbar; or pressing the [Q] key on your keyboard. This opens the **Create iMate** dialog box in Figure 12-35. The **Create iMate** and **Place Constraint** dialog boxes are very similar. You cannot define a transitional iMate in the **Create iMate** dialog box and, because an iMate is only half of a constraint, only one selection button is available in the **Create iMate** dialog box.

Figure 12-35.
The **Create iMate** dialog box, **Assembly** tab.



The first step in creating an iMate using the **Create iMate** dialog box is to select the tab that corresponds to the type of iMate constraint you want to define. You can choose to define assembly and motion iMates by picking the corresponding tab. Again, you select only one component in the **Create iMate** dialog box, instead of selecting two components, as in the **Place Constraint** dialog box.

If the component you insert, or the existing assembly, contains more than one iMate, Inventor searches for iMate matches between the component and the assembly. By default, each possible match is searched based on the order of iMates in the browser. For example, when adding a pin to the assembly of the two plates, each with four insert iMate holes, shown in Figure 12-36, the pin will initially be placed in hole 1 of PLATE:1 because of its position in the browser. If you place additional pins, or search for an alternative location for the pin, hole 2 of PLATE:1 will match next, followed by hole 3 of PLATE:1, with hole 4 of PLATE:2 matched last. An alternative, or addition to the default iMate search pattern is to add a match list when you create an iMate. See Figure 12-35. The match list names you create must correspond to the names of the matching iMates. For example, the match list names for the pin in Figure 12-35 correspond to two of the plate iMates shown in Figure 12-36. When using the match list, the pin will initially be placed in hole 2 of PLATE:1. If you place additional pins, or search for an alternative location for the pin, hole 4 of PLATE:1 will match next, followed by hole 2 of PLATE:2, then hole 4 of PLATE:2. Once all match list iMates have been searched, Inventor will continue to search all other possible iMate matches.

The following example identifies a typical approach to creating iMates for three separate parts. The first part, SHOCK_BODY.ipt, shown in Figure 12-37, contains four iMates. Figure 12-37 shows that many iMates can be added to one component. These iMates have been named using the **Name** edit box in the **Create iMate** dialog box. Providing descriptive iMate names can be helpful when inserting components and referencing iMates. Two or more iMates added to a single component that are used for the same assembly operation can be linked together to form a ***composite iMate***. Composite iMates are used to combine two or more iMates so that when you place a component, you do not have to select multiple iMate constraints. Several constraint operations can be conducted using a single composite iMate. To create a composite

Figure 12-36.
By default, Inventor searches for iMate matches based on the position of assembly iMates in the browser.

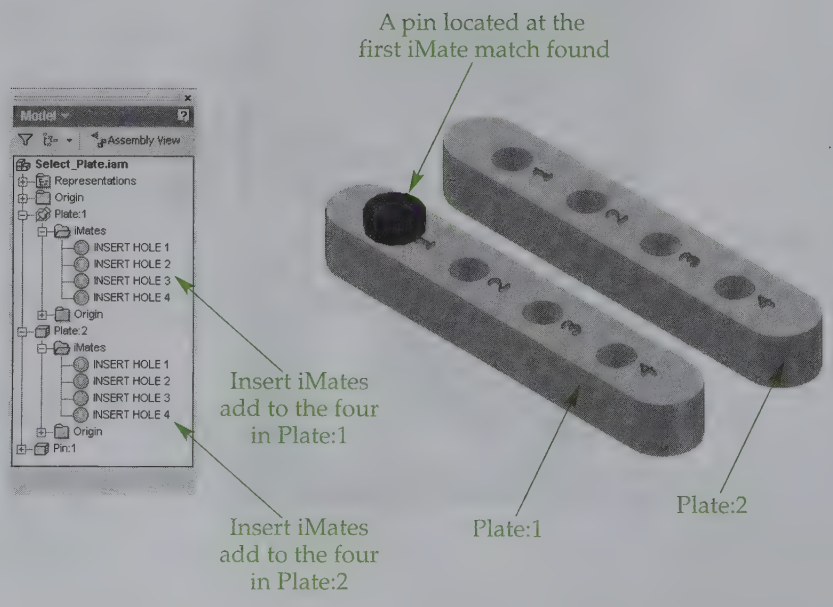
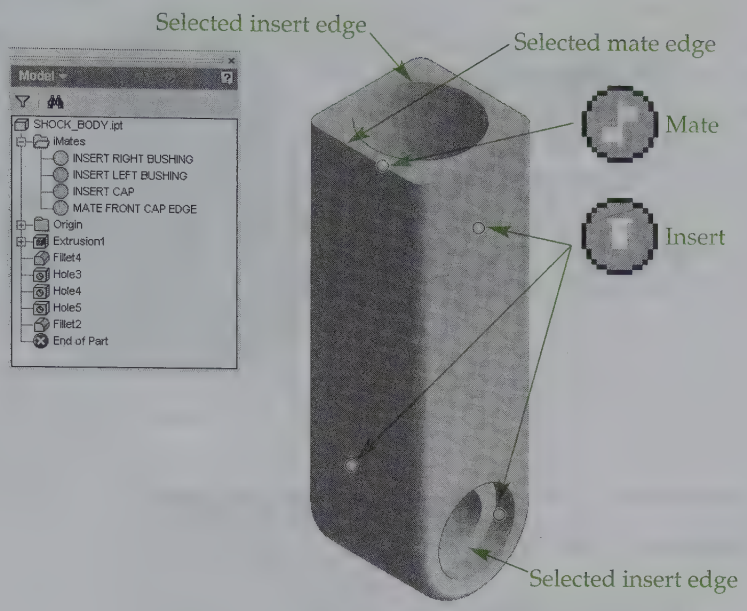


Figure 12-37.
Creating multiple iMates in a single part.



iMate, expand the iMate folder in the browser for the selected component. Then pick each of the iMates in the iMate folder that you want to link together, using the [Ctrl] or [Shift] key on your keyboard. Finally, right-click and select the **Create Composite** pop-up menu option. See Figure 12-38. The iMate symbols specify the type of iMate created and the iMate status. When a constraint between two iMates is identified, the symbol is no longer visible. Next, SHOCK_BODY.ipt is closed and SHOCK _BUSHING.ipt, shown in Figure 12-39, is opened, which contains a single insert iMate. Then, SHOCK_BUSHING.ipt is closed, and SHOCK_CAP.ipt, shown in Figure 12-40, is opened. SHOCK_CAP.ipt requires a composite iMate that contains an insert and mate iMate.

NOTE



Notice only iMates that control constraints between two components are composite. All other iMates remain individual.

Figure 12-38.
Creating a composite iMate.

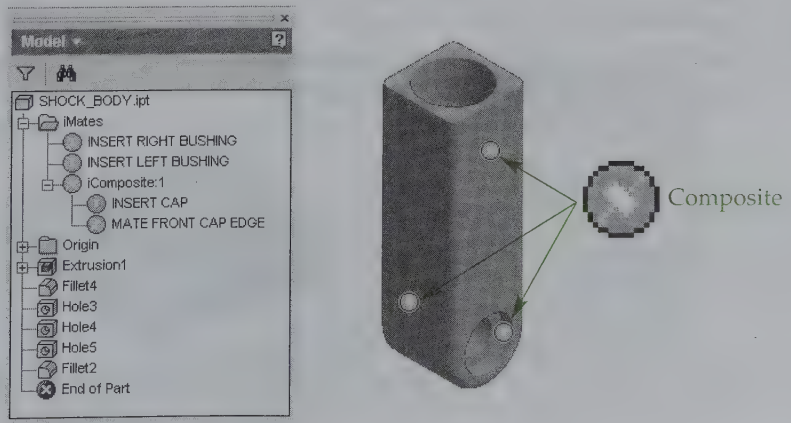


Figure 12-39.
Creating one iMate in a single part.

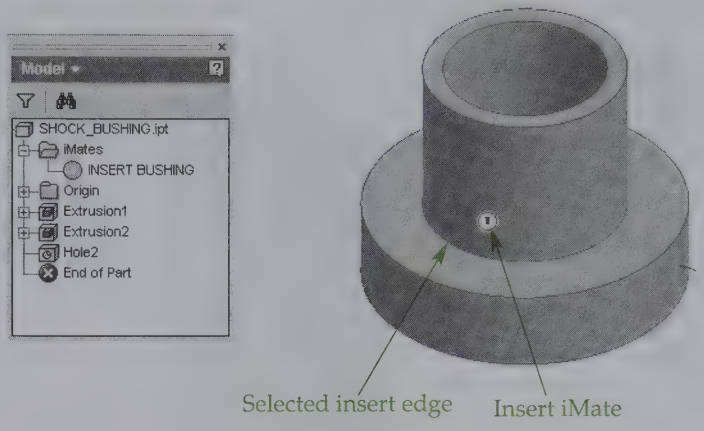
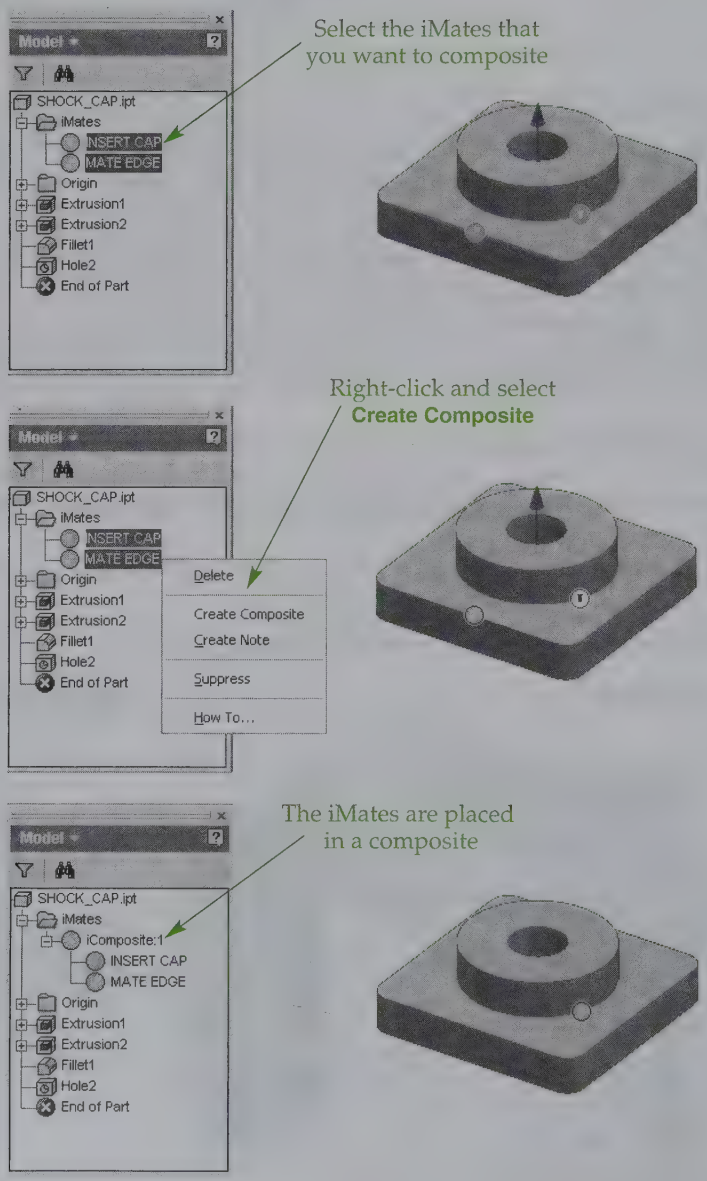
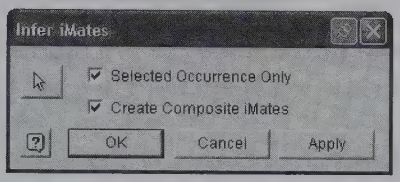


Figure 12-40.
Creating a composite iMate.



iMates can also be generated using existing assembly constraints placed using the constraint techniques previously described. When you generate iMates using mate, angle, tangent, and insert constraints, the existing constraints remain in the assembly file, while iMates are extracted and created in the separate component files. To create iMates using existing assembly constraints, first open an assembly file that contains assembly components and constraints. Next, identify the assembly component from which you want to extract constraints to create iMates. Then, right-click on the component name in the browser or the actual component in the graphics window, and select **Infer iMates...** from the **Component** cascading submenu. Do not try to access the **Infer iMates** tool from actual constraints. You must right-click on the component that contains the constraints to select **Infer iMates...** from the **Component** cascading submenu. When you pick **Infer iMates...**, the **Infer iMates** dialog box is displayed. See Figure 12-41.

Figure 12-41.
The **Infer iMates** dialog box.



By default, the component from which you access the **Infer iMates** dialog box is selected. However, you can extract iMates from a different component by picking the **Select Component** button and choosing a different component name from the browser or the component in the graphics window. When multiple copies of a component are found in the same assembly, pick the **Select Occurrence Only** check box to extract iMates only to the selected component. Unpick the **Select Occurrence Only** check box if you want the iMates added to each component occurrence. If the selected component contains two or more iMates that are used to generate the same assembly, you can select the **Create Composite iMates** check box to link the iMates together and automatically form a composite iMate.

Once you select or deselect the desired check boxes in the **Infer iMates** dialog box, pick the **Apply** button to generate iMates for the selected component based on the assembly constraints. After you pick the **Apply** button, the **Infer iMates** dialog box is still active, allowing you to continue selecting components and adding iMates, if desired. Pick the **OK** button to generate iMates and exit the **Infer iMates** dialog box.

Another option for adding iMates is to automatically infer iMates by selecting the **Infer iMates** check box found in the dialog box of certain part features, including extrusions, revolutions, and holes. When you pick the **Infer iMates** check box, Inventor attempts to infer iMates based on circular feature edges, such as the edge of an extruded circle. The result is typically an insert iMate placed on all possible circular edges. The iMates may or may not be relevant to the assembly.

NOTE



A constraint is divided between two components. As a result, the same constraint is listed under two components in the browser. To locate and highlight the corresponding half of a constraint, right-click on a constraint in the browser and select the **Other Half** pop-up menu option.

PROFESSIONAL TIP

When you move your cursor over a constraint in the browser, an extensive tooltip is displayed. Use these tooltips to help locate and clarify constraint relationships.

Exercise 12-12 Complete the Exercise on the Student CD.

Exercise 12-13 Complete the Exercise on the Student CD.

Exercise 12-14 Complete the Exercise on the Student CD.

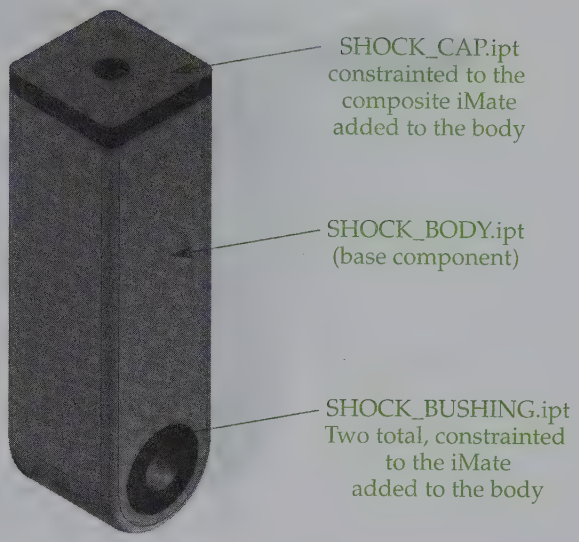
Using iMates

Once you specify component iMates, there are a few different techniques to link the iMates together, depending on the application and design process. The following information describes each method: The most effective way of using iMates to create constraints between components is to use the **Place Component** tool. However, this technique does not work if the components are already inserted in the assembly file. When you use the **Place Component** tool to insert components that contain iMates, the inserted components automatically constrain to each other. The first step in applying this technique is to insert the base component. Then, use the **Place Component** tool to insert additional components. See Figure 12-42. When using this method, you must ensure the **Use iMate** check box is selected in the **Place Component** dialog box. A preview of the iMate constraint is shown. If the preview looks acceptable, pick the location or press [Enter]. You can place additional copies of the component by picking a location in space, or pressing [Enter]. If you only need to place one component, or after you place multiple copies, right-click and select the **Done** menu option, or press [Esc].

Several additional options are available for placing iMate components, depending on the number of iMates matches. Right-click and select one of the following options, or use the shortcut key listed to place iMate components:

- **Use iMates**, [Ctrl]+[I]. This option is selected if you choose the **Use iMate** check box of the **Place Component** dialog box. Deselect the **Use iMates** option to place the component without constraining with iMates.
- **Place at all matching iMates.** Pick this option to place and constrain the component at all possible matching assembly iMate locations. Selecting **Place at all matching iMates** can be effective when the same component must be inserted and constrained multiple times.
- **Skip remaining iMate results**, [Spacebar]. Select this item if you do not want to apply additional iMate constraints, usually when more iMates are available than needed or wanted.

Figure 12-42.
An example of the constrained iMate assembly.



- **Generate remaining iMate results**, [Ctrl]+[Enter]. Choose this option to apply all additional iMate constraints without individually selecting each iMate. Picking **Generate remaining iMate results** will save time when it is apparent that all other iMates can be solved automatically.
- **Restart.** Pick this option to undo iMate selection.
- **Next Component iMate**, [→]. Choose this item to cycle to the selection to the next iMate located on the component you are inserting.
- **Previous Component iMate**, [←]. Pick this option to cycle the selection to the previous iMate located on the component you are inserting.
- **Next Assembly iMate**, [↑]. Select this item to cycle the selection to the next iMate found on the assembly.
- **Previous Assembly iMate**, [↓]. Choose this item to cycle the selection to the previous iMate found on the assembly.
- **Next Instance Assembly iMate**, [Ctrl]+[→]. This option is available if the assembly contains two or more of the same components, and can be used to select the same iMate, but on the next component.
- **Previous Instance Assembly iMate**, [Ctrl]+[←]. This item is available if the assembly contains two or more of the same components, and can be used to select the same iMate, but on the previous component.

The [alt]-drag method can be used to place iMate components when components with iMates have already been inserted into the assembly and are not constrained. To drag-mate two iMate components, hold down the [Alt] key on your keyboard. Then, while still pressing the [Alt] key, hold down the left mouse button on the iMate symbol on the component you want to move into place and drag the iMate symbol to the other iMate symbol on the component to which you want to constrain. Only iMates that can be used for the selected constraint are visible. Once you are satisfied with the intended constraint, release the left mouse button and the [Alt] key to generate the constraint. See Figure 12-43.

Another option for constraining components with iMates that have already been inserted into the assembly and are not constrained is to use the **Place Constraint** tool. The process is similar to using the **Place Constraint** tool to constrain assembly

Figure 12-43.
An example of a drag-mate iMate constraint.



components without iMates. The only difference is the component elements you select. Instead of selecting faces, edges, axes, or points, you select iMates to establish constraints between the components. However, you must select the constraint type that corresponds to the specified iMate.

NOTE

You cannot select composite iMate symbols when using the **Place Constraint** tool. However, you can expand a composite iMate in the browser and select individual iMates in the browser.

Exercise 12-15 Complete the Exercise on the Student CD.

Editing Constraints

Assembly constraints can be edited much like other Inventor elements. To temporarily remove the effects of a constraint, but not actually delete the constraint, right-click on the constraint in the browser and selecting the **Suppress** menu option. Use the **Edit Constraint** dialog box to make changes to a constraint. See Figure 12-44. The **Edit Constraint** dialog box is exactly the same as the **Place Constraint** dialog box, but contains the options used to create the constraint. Access the **Edit Constraint** dialog box by right-clicking on an existing constraint in the browser, and selecting the **Edit** menu option. Although you can use the **Edit Constraint** dialog box to redefine specified constraint offset, angle, ratio, or distance, depending on the selected constraint, the **Value** edit box is often an easier solution. See Figure 12-45. Access the **Value** edit box by picking the desired constraint in the browser. Then enter or select a new value and press the [Enter] key on your keyboard. Constraint values can also be modified using the **Edit Dimension** dialog box, which you can access by double-clicking on a constraint in the browser, or right-clicking on a constraint in the browser and selecting the **Modify** pop-up menu option. Just as with any other **Edit Dimension** dialog box, enter or select a new value in the edit box, and pick the **OK** button to generate the changes.

Figure 12-44.
The **Edit Constraint** dialog box.

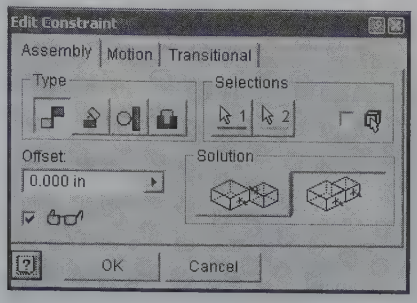
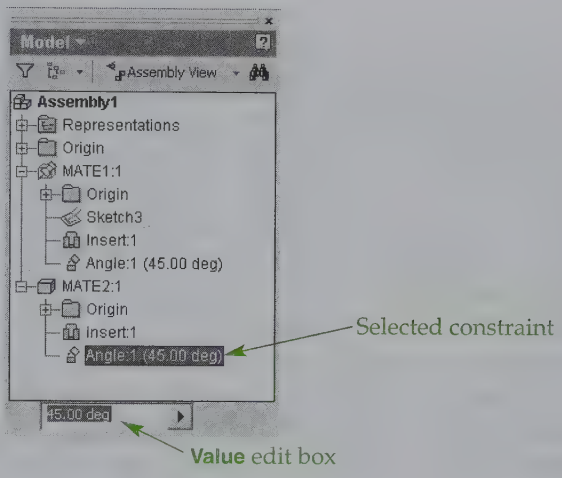


Figure 12-45.
The **Value** edit box.



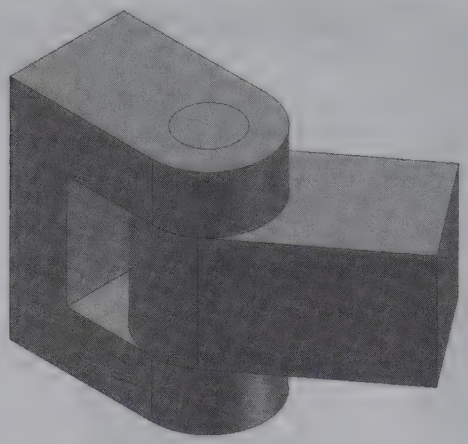
Driving Constraints

One way to view component freedom of movement is to drag a component by selecting it, holding down the left mouse button, and rotating, sliding, or offsetting the component, depending on the design and existing constraints. Although dragging is an effective way to quickly analyze design progress, driving constraints is also valuable, especially for assembly components that may be difficult to drag. You can ***drive*** a constraint to show the amount of movement between components, pause movement, drive adaptivity, and detect collisions between components. A good example of a driven constraint can be seen with the assembly shown in Figure 12-46. In this example, the components can be driven using an angle constraint from a specified angle, such as –110° to another specified angle, such as 95°.

Assuming an acceptable driving constraint is available, typically an angle or mate, you can drive the constraint by right-clicking on the constraint in the browser and selecting the **Drive Constraint** pop-up menu option. When you choose **Drive Constraint**, the **Drive Constraint** dialog box is displayed with the appropriate units for the selected constraint. See Figure 12-47. For example, if you select an angle constraint, units are displayed in degrees. Or if you select a mate constraint, units are displayed in inches or millimeters depending on the design. The following identifies each of the items available in the **Drive Constraint** dialog box:

- **Start.** Use this edit box to enter, select, or measure the desired start point of the drive. By default, the start position is set at the value you specify when you create the constraint, such as the offset or the angle. For example, when

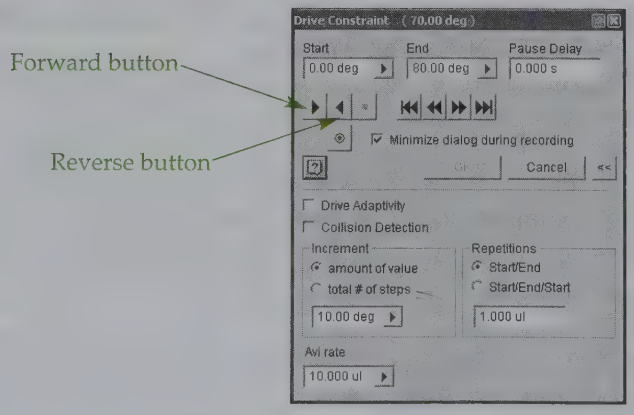
Figure 12-46.
A simple example of a driven constraint application.



working with a mate constraint, if you specify an offset of 5 mm, the start value is 5 mm. Figure 12-47 shows a possible start value for an angle constraint.

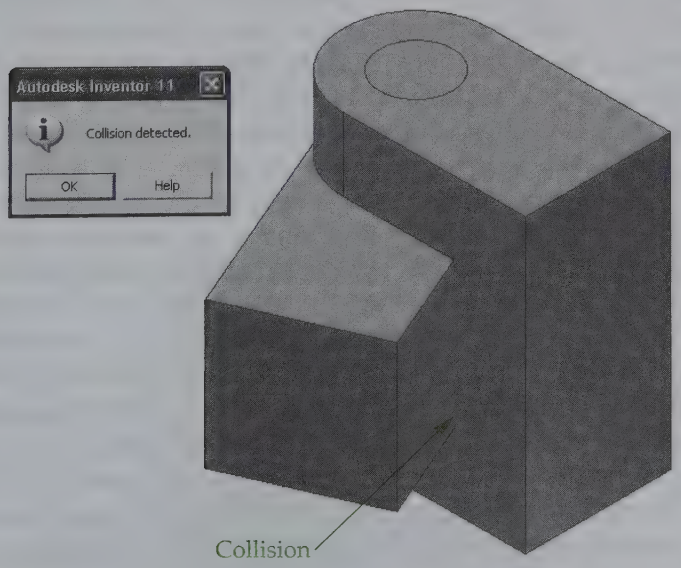
- **End.** This edit box is used to enter, select, or measure the desired end position of the drive. By default the end point is set at the offset or angle value you specify when you create the constraint, plus 10. For example, when working with a mate constraint, if you specify an offset of 5 mm, the start value is 15 mm. Figure 12-47 shows a possible end value for an angle constraint.
- **Pause Delay.** Driven constraints move a certain number of steps. For example, you can set an angle constraint to move 5° per step. Use the **Pause Delay** edit box to specify the number of seconds between steps. The default value is 0, which results in no pause between steps.
- **Forward.** Pick this button to drive the constraint forward. The **Forward** button is available only when both the **Start** and **End** edit boxes have values and the constraint is driven to an appropriate location.
- **Reverse.** Pick this button to drive the constraint in the reverse direction. The **Reverse** button is available only when both the **Start** and **End** edit boxes have values and the constraint is driven to an appropriate location.
- **Stop.** This button is available when a constraint is being driven and allows you to stop the component movement.

Figure 12-47.
The **Start** edit box **Drive Constraint** dialog box.



- **Minimum.** Use this button to automatically return to the start position.
- **Reverse Step.** Pick this button to drive the constraint in the reverse direction, a single step in the movement progression.
- **Forward Step.** Pick this button to drive the constraint in the forward direction, a single step in the movement progression.
- **Maximum.** Use this button to automatically return to the end position.
- **Record.** Pick this button to record the driven constraint movement as an animation file. When you select the **Record** button, the **Open** dialog box appears, allowing you to specify a location for the animation file. You can also choose the file format by selecting either AVI or WMV from the **Save as type:** drop-down list. If you do not want the **Drive Constraint** dialog box displayed while the drive is recorded, pick the **Minimize dialog during recording** check box.
- **Drive Adaptivity.** Pick this check box to drive the adaptive properties of components within relation to the constraint. When you select the **Drive Adaptivity** check box, the **Collision Detection** check box is not available because of the adaptive nature of the drive.
- **Collision Detection.** You can select this check box to automatically identify a collision between the driven constraint and moving components. As shown in Figure 12-48, the components involved in the collision are highlighted, and the collision detection message is displayed. Pick the **OK** button to accept the collision and modify the drive. When the **Collision Detection** check box is selected, the **Drive Adaptivity** check box is not available.
- **Increment.** Use this area to specify the incremental movement of the drive. Pick the **amount of value** radio button if you want to define a certain increment value, such as 10° for a driven angle constraint, or 5" for a driven mate constraint. Then, specify or select the value from the **Increment** edit box. A high increment value results in fewer increments and quicker movement. Pick the **total # of steps** radio button if you want to define a certain number of steps, such as 5 or 10. This means the total amount of drive movement is divided into 5 or 10 steps. Use the **Increment** edit box to specify the number of steps. In contrast to

Figure 12-48.
An example of a drive collision detection application.



the **Amount of value** option, a high number of steps results in more increments and slower movement.

- **Repetitions.** This area allows you to define the repetition options for the drive. Pick the **Start/End** radio button to drive the constraint from the start position to the end position. To cause the drive to return back to the start after it meets the end position, pick the **Start/End/Start** radio button. Then, identify the number of times you want the drive to move from start to end, or from start to end to start, by specifying a value in the **Repetitions** edit box.
- **Avi rate.** This drop-down list is available when you record the drive, and save the animation file in an AVI format. The AVI rate defines the number of increments copied during the animation recording.
- **Apply.** Pick this button to save the drive settings and apply them to the constraint and components. When you finish driving a constraint, you can pick the **Close** or **Cancel** button to exit the tool.

NOTE



While constraints are being driven, you can use tools such as **Rotate**, **Pan**, **Zoom**, and **Look At** to better analyze the design movement.



Complete the Exercise on the Student CD.

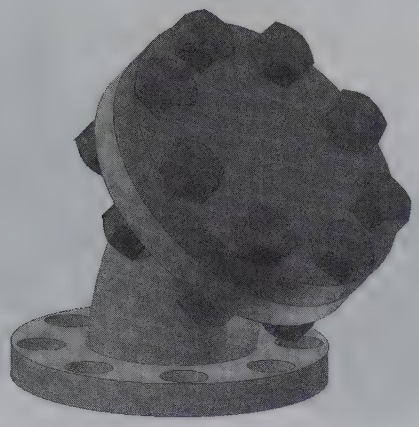
Patterning Components

Often assemblies contain many of the same parts, or even subassemblies, that must be arranged in a certain pattern, such as in Figure 12-49. You can use the tools described earlier to place separate components and then individually constrain the components to other components in a pattern. However, Inventor provides you with the **Pattern Component** tool, which allows you to quickly generate a circular or rectangular pattern from a single component or group of existing components.

A reoccurrence of components, in a designated configuration, is known as a ***component pattern***. Component patterns and the **Pattern Component** tool are very similar to feature patterns and the feature patterning tools available in the part modeling environment. However, the **Pattern Component** tool allows you to associate the assembly component pattern with a feature component pattern. For example, the nuts and bolts patterned in the assembly shown in Figure 12-49 are associated with the holes in the cover plate and elbow. As a result, any changes made to the component with the feature pattern are parametrically translated to the assembly component pattern.

Assuming you have at least one existing component in the assembly file that you want to pattern, such as a screw, and have the corresponding components necessary for generating an associative pattern, you can begin the component patterning process. First, access the **Pattern Component** tool by picking the **Pattern Component** button on the **Assembly Panel** bar; picking the **Pattern Component** button on the **Assembly Panel** toolbar; selecting the **Pattern Component...** option of the **Insert** pull-down menu; or right-clicking on the component name in the browser or the component in the graphics window, and selecting **Pattern** from the **Component** cascading submenu. This opens

Figure 12-49.
An example of an assembly with a circular pattern of components.



the **Pattern Component** dialog box in Figure 12-50. Select one of the following tabs, and use the information provided to create the desired component pattern:

- **Associative.** This tab, shown in Figure 12-50, is initially active when you access the **Pattern Component** dialog box and is probably the most versatile assembly component patterning option. Use the **Associative** tab to pattern an existing component, or group of components, in a rectangular or circular fashion, and in relation to an existing component feature pattern. For example, Figure 12-51 shows a component pattern of screws arranged in a rectangular pattern in reference to the slide bar hinge connector holes.
  To create an associative component pattern, pick the **Select Component** button if it is not already selected. Then choose the component, such as a bolt, or group of components, such as a bolt, washer, and nut, that you want to pattern. The component in Figure 12-51 is an inserted and constrained screw. Once you select the component or components to pattern, pick the **Associative Feature Pattern Select** button, and pick the part feature pattern to define the assembly component pattern. The associative feature pattern in Figure 12-51 is the pattern of holes on the slide bar hinge connector. When you select the associative feature pattern, the name of the pattern is shown in the **Feature Pattern Select** display box, and a

Figure 12-50.
The **Pattern Component** dialog box, **Associative** tab.

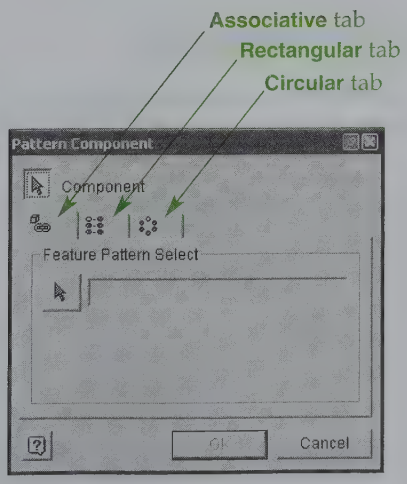
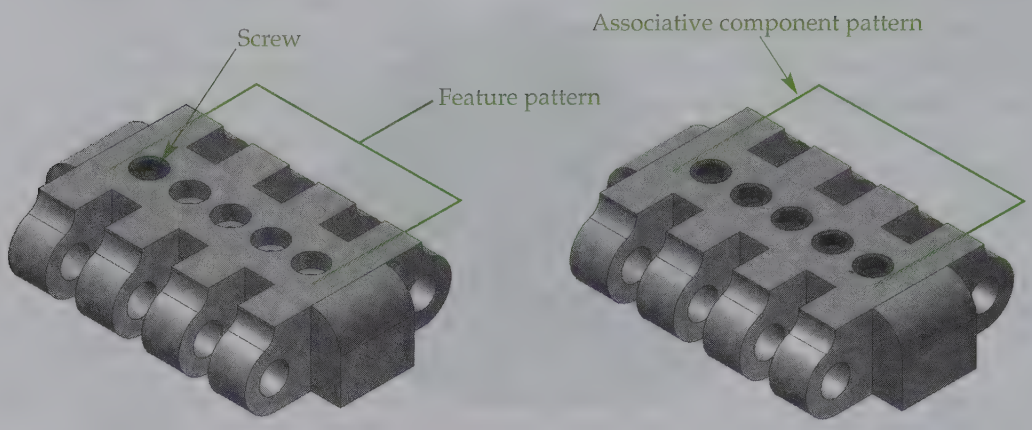


Figure 12-51.
This slide bar hinge connector component contains feature patterns and the assembly contains an associative component pattern.



preview of the component pattern is displayed. If the correct feature pattern and preview is shown, pick the **OK** button to generate the component pattern.

- **Rectangular.** This tab, shown in Figure 12-52, allows you to create a rectangular pattern of a component or series of components, without identifying an associative feature pattern. For example, Figure 12-53 shows a rectangular component pattern of three brackets, which is later linked with other components created in-place. Generating a rectangular component pattern is very similar to creating a rectangular feature pattern.

  To create a rectangular component pattern, pick the **Select Component** button if it is not already selected. Then, choose the component, or group of components, that you want to pattern. The component in Figure 12-53 is an inserted and grounded bracket. Once you select the component or components to pattern, pick the **Column Direction** button and select the direction of the column of patterned components. You can select any feature edge, axis, or work axis available in the model, including the default reference axes located in the browser bar Origin folder. When you choose a direction, an arrow shows you the specified pattern path, and a preview displays the pattern operation. If the direction is not correct, pick the **Flip** button. The next step is to specify how many copies of the component or series of components you want to create by entering or selecting a value in the **Count** edit box. Then, define the distance

Figure 12-52.
The **Pattern Component** dialog box, **Rectangular** tab.

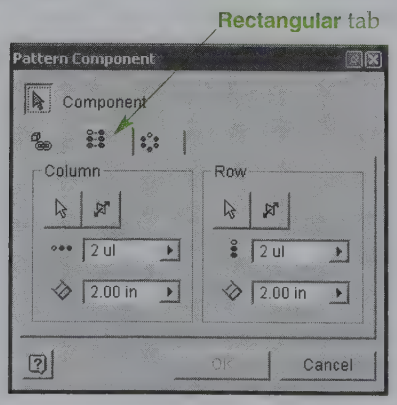
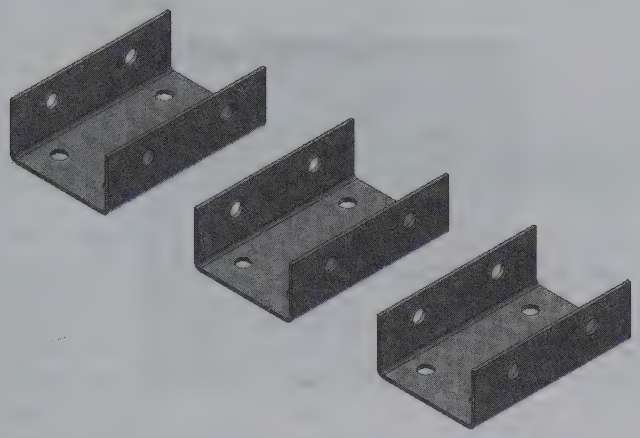


Figure 12-53.
An example of a rectangular component pattern.



between the copies by entering or selecting a value from the **Spacing** edit box. Spacing is measured by the width of the selected features and the distance between the copies. It is not just the space between features. For example, if you want to pattern a block that is 1″ wide, and you want a 1/2″ space between pattern copies, you must specify a 1 1/2″ spacing.

Once you fully define the **Column Direction**, repeat the steps previously described for placing rows. Pick the **Row Direction** button, and select the desired direction. Again, you can select any component edge, axis, or work axis available in the model, including the default reference axes located in the browser bar Origin folder. Theoretically, you can choose the same edge or axis you chose for the column direction, or a parallel edge or axis. However, for most applications, you should select an edge or axis that is perpendicular, or at least nonparallel, to the first direction. When you choose the row direction, an arrow shows you the specified pattern path, and a preview displays the pattern operation. After you define the row direction, specify how many copies of the component or series of components you want to create by entering or selecting a value in the **Count** edit box. Then define the distance between the copies by entering or selecting a value from the **Spacing** edit box. Again, spacing is measured by the width of the selected components and the distance between the copies. It is not just the space between components. Finally, pick the **OK** button to generate the rectangular pattern.

- **Circular.** This tab, shown in Figure 12-54, allows you to create a circular pattern of a component or series of components, without identifying an associative feature pattern. For example, Figure 12-55 shows a component pattern of six studs arranged in a circular pattern, which, when fabricated, are welded to the plate. Generating a circular component pattern is very similar to creating a circular feature pattern.

  To create a circular component pattern, pick the **Component** button if it is not already selected. Then choose the component, or group of components, that you want to pattern. The component in Figure 12-55 is an inserted and constrained stud. Once you select the component or components to pattern, pick the **Axis Direction** button and select the axis about which the patterned components are generated around. You can select any existing component or work axis, including the work axes available in the browser bar **Origin** folder. When you choose a direction, an arrow and axis shows you the specified pattern path, and a preview displays the pattern operation. If the direction is not correct, pick the

Figure 12-54.
The **Pattern Component** dialog box, **Circular** tab.

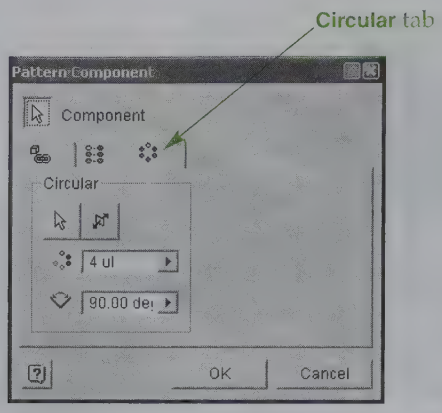
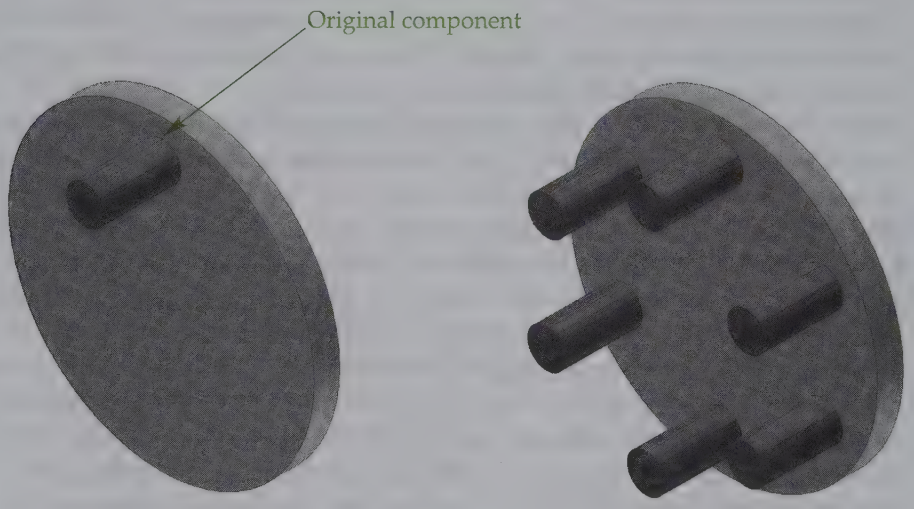


Figure 12-55.
A circular pattern of the original component was created in this assembly.

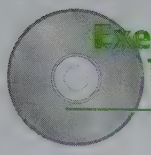


Flip button. The next step is to specify how many copies of the component or series of components you want to create, by entering or selecting a count value in the **Count** edit box. Then define the spacing between the components by entering or selecting an angle in the **Angle** edit box. For example, if you want the copies to be equally distributed around a 360° circle, divide 360 by the count to identify the angle. Finally, pick the **OK** button to generate the circular pattern.

NOTE



To turn off, or hide, certain occurrences of patterned components, expand the component pattern in the browser, right-click the desired occurrence, and pick the **Suppress** menu option. Just as with other assembly components, you can demote component pattern elements, other than the parent element. You can separate a component pattern element from the pattern by right-clicking on the element in the browser and selecting the **Independent** pop-up menu option.



Complete the Exercise on the Student CD.

Mirroring Components

A ***mirrored component*** consists of one or more parts or subassemblies that are mirrored over a specified plane. Mirrored assembly components are typically used to create symmetrical assemblies. Mirroring components can be effective for many applications because it allows you to create a reversed copy of parts or subassemblies, without having to develop additional components. Figure 12-56 shows an example of a subassembly created by mirroring parts.

Assuming you have at least one part or subassembly in the assembly that you want to mirror, the first step to mirroring components is to establish a mirror plane. The ***mirror plane*** acts as the line of symmetry, defining where the specified features are mirrored across. A mirror plane can be any available plane, including planar faces, work planes, and the default reference planes available in the browser. Often, the most effective mirror planes are created using the **Work Plane** tool described earlier in this text. In fact, mirroring components across a work plane is very useful for developing reference geometry that can then be used to generate additional assembly items. For example, the upper bracket shown in Figure 12-56 was created by referencing the symmetrically mirrored legs. Once you identify a mirror plane, you are ready to continue the mirroring process. Mirrored components are created using the **Mirror Components** tool by picking the **Mirror Components** button on the **Assembly Panel** bar; selecting the **Mirror Components** button on the **Assembly Panel** toolbar; or right-clicking on the component name in the browser or the component in the graphics window, and selecting **Mirror** from the **Component** cascading submenu. This opens the **Mirror Components** dialog box in Figure 12-57. To mirror components, pick the **Mirror Plane** button if it is not already active. Then select the mirror plane you identified before

Figure 12-56.
An example of mirroring assembly components.

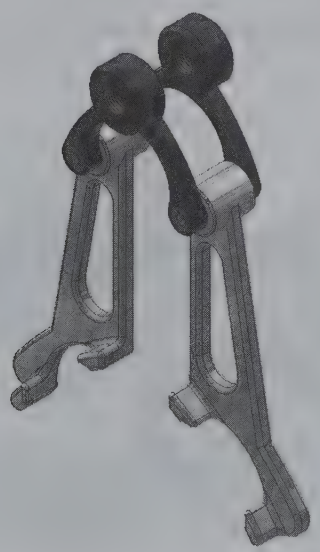
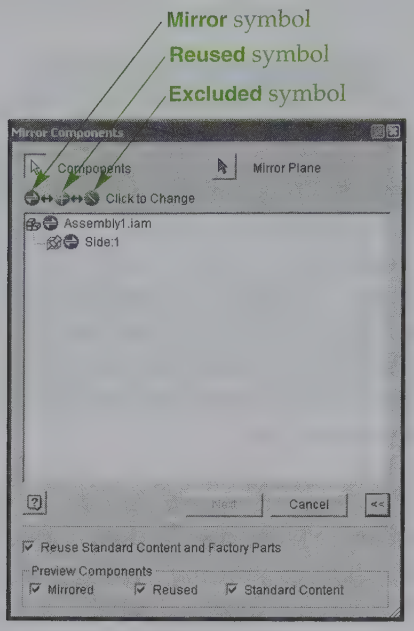


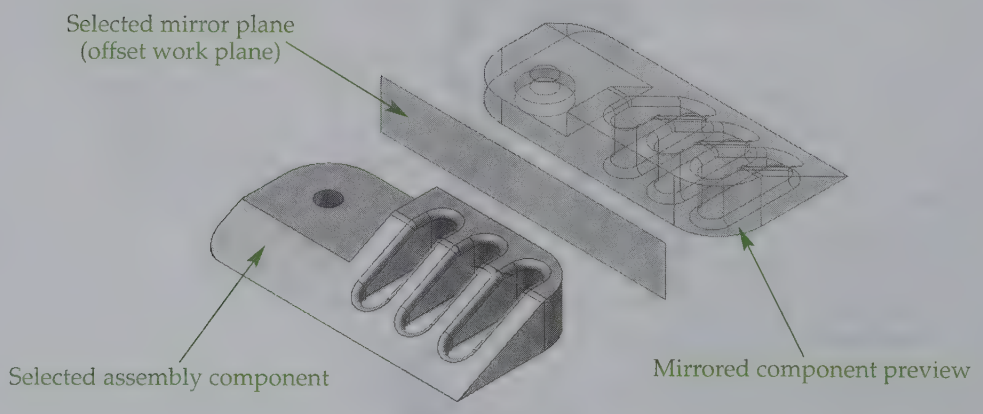
Figure 12-57.
The **Mirror Components** dialog box.



accessing the **Mirror Components** tool from the graphics window or browser. Again, the selected mirror plane can be a planar face, work plane, or default reference plane. When you choose a mirror plane, the **Components** button should become active allowing you to select parts and subassemblies from the graphics window or browser. See Figure 12-58.

As shown in Figure 12-57, all selected components are listed under the assembly file. By default, all the components are set to mirror across the mirror plane as identified by the **Mirror** symbol. Mirroring components using the **Mirror Components** tool creates separate files and places the mirrored items in the files as derived bodies. As a result, in some situations you may not want certain components to be mirrored, but would like the components to be included in the operation. For example, if you want to use a standard bolt in a mirrored assembly, you do not need to create a new mirrored copy of the bolt. For this type of situation, you can choose to reuse a component

Figure 12-58.
Selecting a mirror plane and components.



instead of mirroring the item, which is similar to placing an additional copy of the component into the assembly file. To reuse components, select the **Mirror** symbol to change it to a **Reuse** symbol. If you do not want certain components to be mirrored or reused, pick the **Mirror** symbol until you see the **Excluded** symbol.

NOTE

When you choose to mirror, reuse, and exclude components from the operation, the **Mixed Reuse/Excluded** symbol should be displayed next to the assembly.

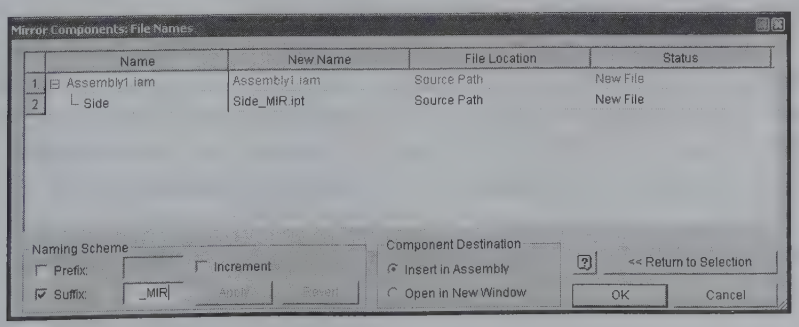
Once you select components and define the mirror status for each selection, you may want to pick the **More** button to access the following additional mirroring preferences and preview options:

- **Reuse Content Library Components and Factory Parts.** Select this check box if you do not want library components, such as shared fasteners or springs, to be mirrored. When this check box is selected, library components are reused, as previously described.
- **Preview Components.** This area contains check boxes that allow you to define the type of components that will be previewed in the graphics window during the mirror process. You can choose to preview mirrored, reused, and standard content library components by selecting the corresponding check box.

The next step to mirroring components is to pick the **Next** button to access the **Mirror Components: File Names** dialog box. See Figure 12-59. As previously mentioned, mirroring components using the **Mirror Components** tool creates separate component files and places the mirrored items in the files as derived bodies. For example, when you mirror a part, a separate part file containing a derived body is created. The **Mirror Components: File Names** dialog box allows you to define file name and location preferences for the new component files created when components are mirrored using the following options:

- **Name.** This column lists each of the components you selected to mirror nested in the parent assembly file.
- **New Name.** Use this column to modify the default name of the new files created as a result of mirroring components. Select the name you want to change and enter a new name in the edit box.
- **File location.** This column identifies where each new file is placed. **Source Path** represents the location of the existing component, but you can create the new file in your Workspace by right-clicking on **Source Path** and selecting the **Workspace** pop-up menu option. You also have the option of placing the new file in a completely different path by right-clicking on **Source Path** and selecting the **User Path** pop-up menu option, followed by picking the path and selecting the **Browse** button to specify a location.

Figure 12-59.
The **Mirror Components: File Names** dialog box.



- **Status.** This column identifies the status of the mirror process based on the specified new file name and location. A **New File** status identifies that the new file name is acceptable and does not already exist. A **Reuse Existing** status, displayed with a yellow background, identifies that you entered a new part file name that is the same as an existing file name, but it can be used. If you enter a new name for an assembly file that is the same as an existing assembly file name, an unacceptable **Name Conflict** status occurs and the background is red.
- **Naming Scheme.** Use this area to specify default new file name characteristics. Select the **Suffix** check box to add the characters defined in the **Suffix** edit box, after the existing file name. For example, a new file based on PART.ipt is named PART_MIR.ipt if the suffix is _MIR. Select the **Prefix** check box to add the characters defined in the **Prefix** edit box, after the existing file name. For example, a new file based on PART.ipt is named MIR_PART.ipt if the prefix is MIR_. When you redefine the prefix and suffix settings, pick the **Apply** button to observe the changes in the columns. To undo the settings and return to the previous naming scheme, pick the **Revert** button. You can also pick the **Increment** check box to apply increments to file names with numbers.
- **Component Destination.** This area allows you to define where the new mirrored components are placed. Select the **Insert in Assembly** radio button to place the mirrored components into the existing, parent assembly file. Or, select the **Open in New Window** radio button to create a new assembly file, and add only the mirrored components to the new file.
- **Return to Selection.** Pick this button to reopen the **Mirror Components** dialog box and redefine your selections.

Finally, pick the **OK** button to generate the mirrored components and separate component files.

NOTE



Mirrored assembly components are created as derived bodies. For more information regarding derived components, refer to the derived components section of this book. Mirrored components are grounded in space, but contain no other assembly constraints.



Complete the Exercise on the Student CD.

Copying Components

In most situations, when you want to add additional copies of existing assembly components to your model, you place the components using one of the component insertion techniques described in this chapter. For example, if your assembly currently contains a part named BOLT, and you want to add another BOLT to the assembly, you can drag BOLT from the browser and drop it into the graphics window. However, in some cases you may want to create a copy of a component but use a different file name, such as BOLT_CPY, and disassociate the new component from the existing component. Copying components in this manner can be effective for many applications

because it allows you to quickly create an exact copy of parts or subassemblies, which can then be modified without affecting the original components.

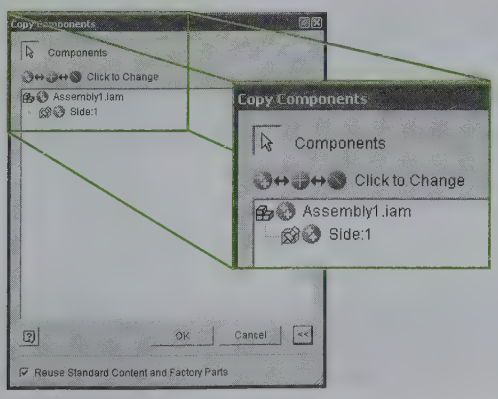
> **NOTE**
>
> Do not confuse the process of inserting, or reusing, existing component files into an assembly, with copying components using the **Copy Components** tool. When you copy a component, a completely new file is created.

Creating new files using existing assembly components is effectively accomplished using the **Copy Components** tool by picking the **Copy Components** button on the **Assembly Panel** bar; selecting the **Copy Components** button on the **Assembly Panel** toolbar; or right-clicking on the component name in the browser or the component in the graphics window, and selecting **Copy** from the **Component** cascading submenu. This opens the **Copy Components** dialog box in Figure 12-60. To copy components, pick the **Components** button if it is not already active. Then, select parts and subassemblies from the graphics window or browser. As shown in Figure 12-60, all selected components are listed under the assembly file. By default, all the components are set to copy as identified by the **Copy** symbol. Copying components using the **Copy Components** tool creates separate files and places the copied items in the files as part or assemblies, depending on whether the components are parts or subassemblies. As a result, in some situations you may not want certain components to be copied, but would like the components to be included in the operation. For example, if you want to use a standard bolt in an assembly, you do not need to create a new copy of the bolt. For this type of situation, you can choose to reuse a component instead of copying the item, which is just like placing an additional copy of the component into the assembly file. To reuse components, select the **Copy** symbol to change to a **Reuse** symbol. If you do want certain components to be copied or reused, pick the **Copy** symbol until you see the **Excluded** symbol.

> **NOTE**
>
> When you choose to copy, reuse, and exclude components from the operation, the **Mixed Reuse/Excluded** symbol should be displayed next to the assembly.

Once you select components and define the copy status for each selection, you may want to pick the **More** button to access the **Reuse Content Library Components**

Figure 12-60.
The **Copy Components** dialog box.

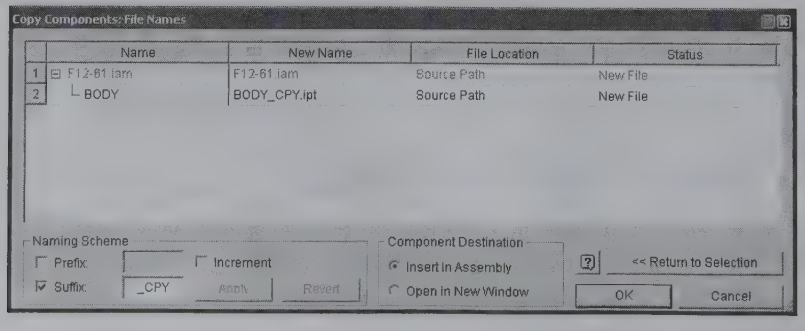


and Factory Parts check box. Select this check box if you do not want library components, such as shared fasteners or springs, to be copied. When this check box is selected, library components are reused, as previously described.

The next step to copying components is to pick the **OK** button to access the **Copy Components: File Names** dialog box. See Figure 12-61. As previously mentioned copying components using the **Copy Components** tool creates separate component files and places the copied items in the files as new parts or assemblies. The **Copy Components: File Names** dialog box allows you to define file name and location preferences for the new component files created when components are copied using the following options:

- **Name.** This column lists each of the components you selected to copy nested in the parent assembly file.
- **New Name.** Use this column to modify the default name of the new files created as a result of copying components. Select the name you want to change and enter a new name in the edit box.
- **File location.** This column identifies where each new file is placed. **Source Path** represents the location of the existing component, but you can create the new file in your Workspace by right-clicking on **Source Path** and selecting the **Workspace** pop-up menu option. You also have the option of placing the new file in a completely different path by right-clicking on **Source Path** and selecting the **User Path** pop-up menu option, followed by picking the path and selecting the **Browse** button to specify a location.
- **Status.** This column identifies the status of the copy process based on the specified new file name and location. A **New File** status identifies that the new file name is acceptable and does not already exist. A **Reuse Existing** status, displayed with a yellow background, identifies that you entered a new part file name that is the same as an existing file name, but it can be used. If you enter a new name for an assembly file that is the same as an existing assembly file name, an unacceptable **Name Conflict** status occurs and the background is red.
- **Naming Scheme.** Use this area to specify default new file name characteristics. Select the **Suffix** check box to add the characters defined in the **Suffix** edit box, after the existing file name. For example, a new file based on PART.ipt is named PART_CPY.ipt if the suffix is _CPY. Select the **Prefix** check box to add the characters defined in the **Prefix** edit box, before the existing file name. For example, a new file based on PART.ipt is named CPY_PART.ipt if the prefix is CPY_. When you redefine the prefix and suffix settings, pick the **Apply** button to observe the changes in the columns. To undo the settings, and return to the previous naming scheme, pick the **Revert** button. You can also pick the **Increment** check box to apply increments to file names with numbers.
- **Component Destination.** This area allows you to define where the new copied components are placed. Select the **Insert in Assembly** radio button to place the

Figure 12-61.
The **Copy Components: File Names** dialog box.



copied components into the existing, parent assembly file. Or, select the **Open in New Window** radio button to create a new assembly file, and add only the copied components to the new file.

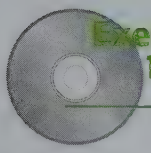
- **Return to Selection.** Pick this button to reopen the **Mirror Components** dialog box and redefine your selections.

Finally, pick the **OK** button to reenter the assembly work environment, and pick a location in space to place the copied assembly component.

NOTE



Copied components are grounded in space according to the location you select, but contain no other assembly constraints.



Exercise 12-19 Complete the Exercise on the Student CD.

Adjusting Components

Once you develop assemblies by inserting, creating, and constraining components, the assembly can be modified using a number of tools and techniques. You can edit the actual part or subassembly component file by opening the file and modifying sketches, features, and other model specifications. To open an individual component file from inside the assembly, right-click on the component name in the browser or the actual component in the graphics window and select the **Open** menu option. Components can also be modified directly in the assembly file by right-clicking on the component name in the browser or the actual component in the graphics window, and selecting the **Edit** menu option; double-clicking on the component name in the browser or the actual component in the graphics window; or selecting the **Modeling View** option in the browser.

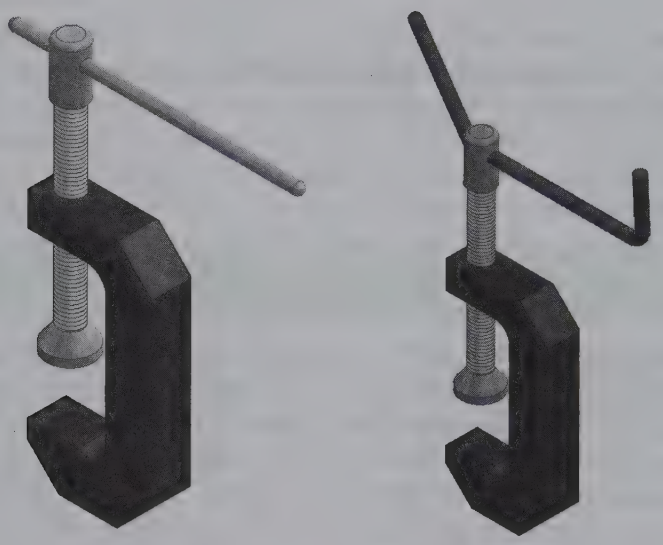
Replacing Components

In many applications, a new or modified component design is developed for existing assemblies. Often, the new component is similar to the old component and may even be constrained the same way. Although you can delete the old component, insert the new component, and constrain the new component to other assembly elements, an easier solution is to replace the old component. Inventor provides the **Replace Component** and **Replace All** tools, which allow you to replace a single component or each occurrence of the same component, with a different component. For example, if an assembly contains eight of the same 1″ (25.4 mm) bolts, you can replace one of the 1″ bolts with a 2″ (49 mm) bolt, or you can replace all the 1″ bolts with 2″ bolts.

To replace a single component, access the **Replace Component** tool by picking the **Replace Component** button on the **Assembly Panel** bar (contained in a flyout button); picking the **Replace Component** button on the **Assembly Panel** toolbar (contained in a flyout button); selecting the **Replace** option of the **Edit** pull-down menu; right-clicking on the component name in the browser or the component in the graphics window, and selecting **Replace** from the **Component** cascading submenu; pressing [Ctrl]+[H]; or picking the **Replace Component** button on the **Assembly Replace** toolbar.

When you access the **Replace Component** tool, you must first select the component you want to replace. If you used one of the right-click access techniques, the component is already selected. Once you pick the desired component, the **Open** dialog box is displayed, allowing you to locate the component you want to use to replace the old component. The replacement component changes to the specified color and is positioned in the same location as the old component. However, if the new component geometry is significantly different than the old, as shown in Figure 12-62, constraints may no longer be established.

Figure 12-62.
An example of a very different replaced component, with constraints automatically removed on placement.



To replace all the instances of a component, such as six of the same bolts, access the **Replace All** tool by picking the **Replace All** button on the **Assembly Panel** bar (contained in a flyout button); picking the **Replace All** button on the **Assembly Panel** toolbar (contained in a flyout button); right-clicking on the component name in the browser or the component in the graphics window, and selecting **Replace All** from the **Component** cascading submenu; selecting the **Replace All** option of the **Edit** pull-down menu; pressing [Shift]+[H]; or picking the **Replace All** button on the **Assembly Replace** toolbar.

When you access the **Replace All** tool, you must first pick the component you want to replace. Once the desired component is selected, the **Open** dialog box is displayed, allowing you to locate the component you want to use to replace the old components. The replacement components change to the specified color, and are positioned in the same location as the old components. However, if the new component geometry is significantly different than the old, constraints may no longer be established.



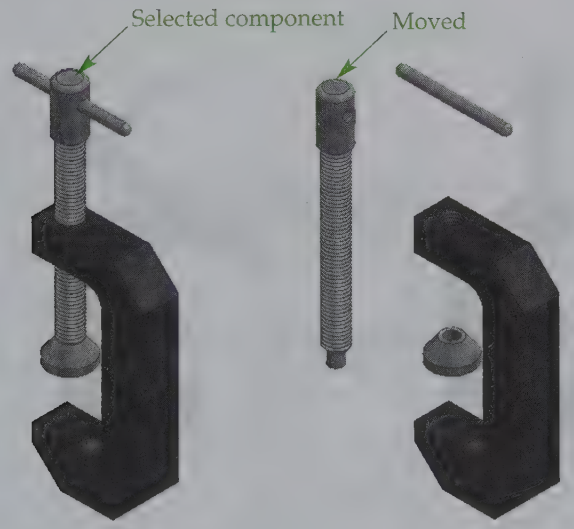
Exercise 12-20 Complete the Exercise on the Student CD.

Moving Components

You are already familiar with the ability to select and drag a component to observe freedom of movement, assuming certain constraints do not allow movement. You also have the option of driving constraints to view component movement. When you move a component that is not constrained, the component remains in the moved location, unless you move it back to the old location, undo the move, or add constraints. Use the **Move Component** tool to move a fully constrained, partially constrained, or unconstrained component anywhere in space. The **Move Component** tool is typically used to move components for clarity and display purposes. Even grounded components can be moved using this tool. To access the **Move Component** tool, pick the **Move Component** button on the **Assembly Panel** bar; pick the **Move Component** button on the **Assembly Panel** toolbar; right-click on the component name in the browser or the component in the graphics window, and select **Move** from the **Component** cascading submenu; or press the [V] key on your keyboard.

When you access the **Move Component** tool, pick and hold down the left mouse button on the component you want to move and drag it to the desired location. Release the left mouse button to drop the component in the new position. Continue moving components as required. Then, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, pick anywhere in space, or access a different tool to exit the **Move Component** tool. See Figure 12-63. When you update the assembly, the move is overridden if constraints are present, bringing components back to their original location in space and in reference to other components. For example, the moved screw in Figure 12-63 returns to its original position when the assembly is updated. However, unconstrained components remain in the moved position, and grounded components are grounded to the moved location.

Figure 12-63.
Moving a component.

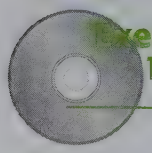


Rotating Components

When you rotate components using the view **Rotate** tool, you are rotating the entire model and work environment in space and do not actually change the model position. However, in contrast, you can rotate a single component, changing its position in space and in relation to other components. When you rotate a component that is not constrained, the component remains in the rotated position, unless you rotate it

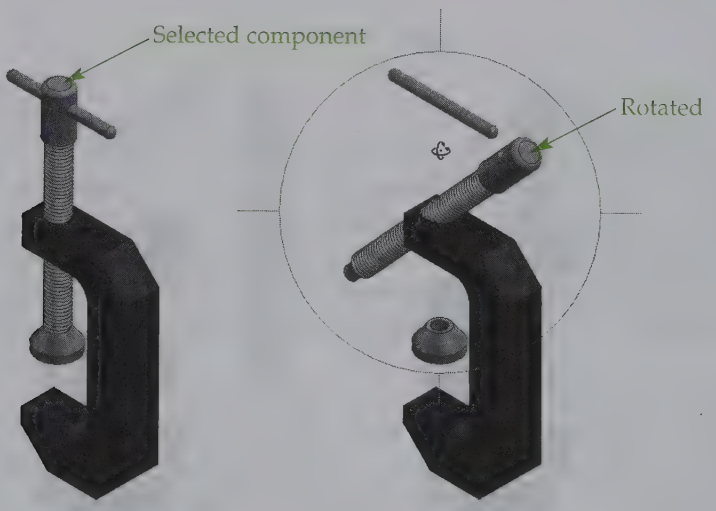
back to the old location, add constraints, or undo the rotate. Use the **Rotate Component** tool to rotate a fully constrained, partially constrained, or unconstrained component, anywhere in space. Like the **Move Component** tool, the **Rotate Component** tool is typically used to rotate components for clarity and display purposes. Even grounded features can be rotated using this tool. To access the **Rotate Component** tool, pick the **Rotate Component** button on the **Assembly Panel** bar; pick the **Rotate Component** button on the **Assembly Panel** toolbar; right-click on the component name in the browser or the component in the graphics window, and select **Rotate** from the **Component** cascading submenu; or press the [G] key on your keyboard.

Once you access the **Rotate Component** tool, pick the component you want to rotate. From this point on, rotating a single component is just like rotating an entire model using the **Rotate** tool. To select another component to rotate while still in the **Rotate Component** tool, position the **Rotate** icon over the desired component, and pick. Continue rotating components as required, and when finished, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, pick anywhere in space, or access a different tool to exit the **Rotate Component** tool. See Figure 12-64. When you update the assembly, the rotate is overridden if constraints are present, bringing components back to their original position in space and in reference to other components. For example, the rotated screw in Figure 12-64 returns to its original position when the assembly is updated. However, unconstrained components remain in the rotated position, and grounded components are grounded to the rotation.



Exercise 12-21 Complete the Exercise on the Student CD.

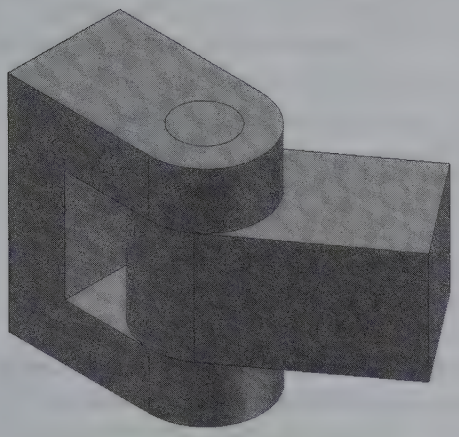
Figure 12-64
Rotating a component.



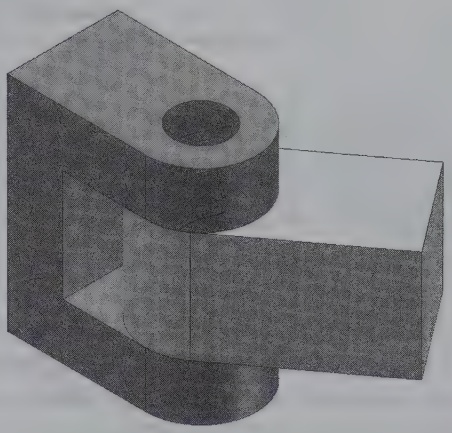
Changing Component Colors

Assembly components are created and assembled using the color of the selected material or another specified color defined by material or color style. Therefore, assembly components made from the same material are typically the same color. Even assembly components made from different materials can have a similar color. As shown in Figure 12-65, it is difficult, and sometimes impossible, to distinguish one component

Figure 12-65.
Components in an assembly are easier to distinguish when they are different colors.



All components are the same color

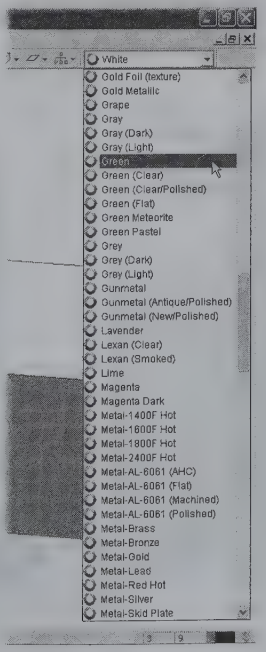


All components are different colors

from another. Fortunately, you change the color of a component without changing material properties. Changing component colors is almost necessary for large assemblies with multiple components, but can even be important for small assemblies as shown in **Figure 12-65.** When component colors are modified to contrast other components, designs become clearer and easier to understand.

To change the color of a component while in an assembly file, select the component in the graphics window or the component name in the browser. Then, select the desired color from the **Color** drop-down list available in the **Inventor Standard** toolbar. See **Figure 12-66.** If the assembly components are made out of different materials, with colors that already contrast, pick the **As Material** color option, which displays the color of the component as the specified material color.

Figure 12-66.
The **Color** drop-down list.



NOTE

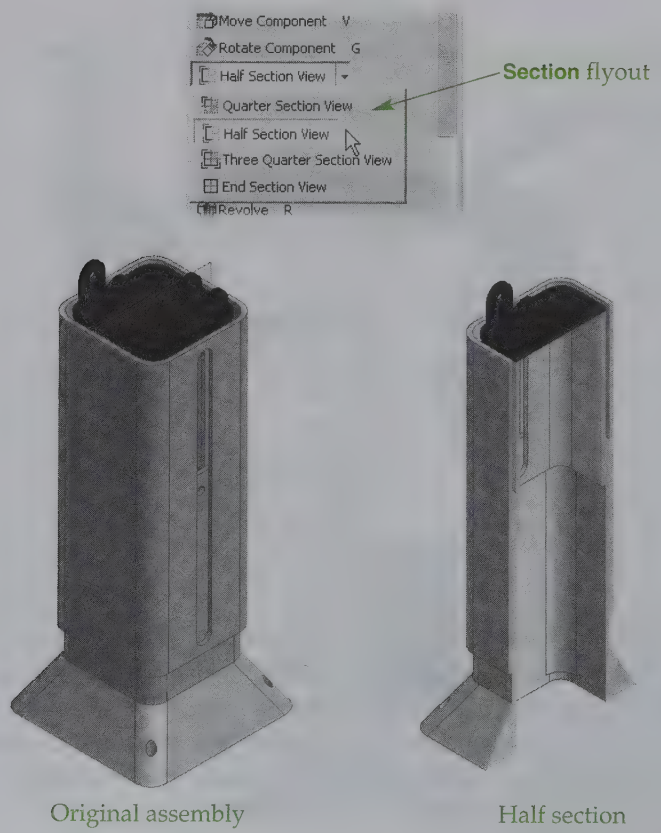
For an alternative coloring option, you can slightly modify actual material colors and leave component colors as **As Material**.

Exercise 12-22 Complete the Exercise on the Student CD.

Sectioning Assemblies

Assemblies are often so complex that it is difficult to clearly observe a design, visualize assembly component relationships, or access component elements when constraining existing components or creating components in-place. As a result, a number of assembly sectioning tools are available. Much like 2D drawing sections, assembly model sections cut away material to display hidden component features. See Figure 12-67. Depending on the application and the amount of material you want to remove, you can create a quarter section view, a half section view, or a three-quarter section view. Assembly section views are created by selecting the appropriate existing face, faces, work plane, or work planes, depending on the type of section view. Access the desired assembly sectioning tool using one of these options: pick the **Quarter Section View**, **Half Section View**, or **Three-Quarter Section View** tool button from the **Section View** flyout button on the **Assembly Panel** bar; pick the **Quarter Section View**, **Half Section View**, or **Three-Quarter Section View** tool button

Figure 12-67.
Select the **Section** flyout to start creating a half section view of the assembly.



from the **Section View** flyout button on the **Assembly Panel** toolbar; or pick the **Quarter Section View, Half Section View,** or **Three-Quarter Section View** tool button on the **Assembly Section** toolbar.

Once you access one of the available section view tools, use the following information to create the section:

- A ***quarter section*** removes all but one quarter of material from two intersecting faces or planes. Once you access the **Quarter Section View** tool, create the section by picking two intersecting component faces or work planes. In addition, the side of the face or plane you select defines which quarter of material remains. See Figure 12-68. If the section looks acceptable, right-click and select the **Done** menu option, press [Esc], or access a different tool to exit the **Quarter Section View** tool. If the section is not correct, you may be able to solve the problem by right-clicking and selecting the **Flip Section** menu option to change the section of removed material, or pick the **Three Quarter Section View** option to define a three-quarter section view.
- A ***half section*** removes one-half of the material from a selected face or plane. Once you access the **Half Section View** tool, create the section by picking the desired component face or work plane. As with quarter section views, the side of the face or plane you select defines which half of the material is removed. See Figure 12-69. If the section looks acceptable, right-click and select the **Done** menu option, press [Esc], or access a different tool to exit the **Half Section View** tool. If the section is not correct, you may be able to solve the problem by right-clicking and selecting the **Flip Section** menu option to change the section of removed material.
- A ***three-quarter section*** removes only one quarter of material from two intersecting faces or planes. Once you access the **Three Quarter Section View** tool, create the section by picking two intersecting component faces or work planes. Again, the side of the face or plane you select defines which quarter of material is removed. See Figure 12-70. If the section looks acceptable, right-click and select the **Done** menu option, press [Esc], or access a different tool to exit the **Three Quarter Section View** tool. If the section is not correct, you may be able to solve the problem by right-clicking and selecting the **Flip Section** menu option to change the section of removed material, or pick the **Quarter Section View** option to define a quarter section view.

Figure 12-68.
Creating a quarter section view.

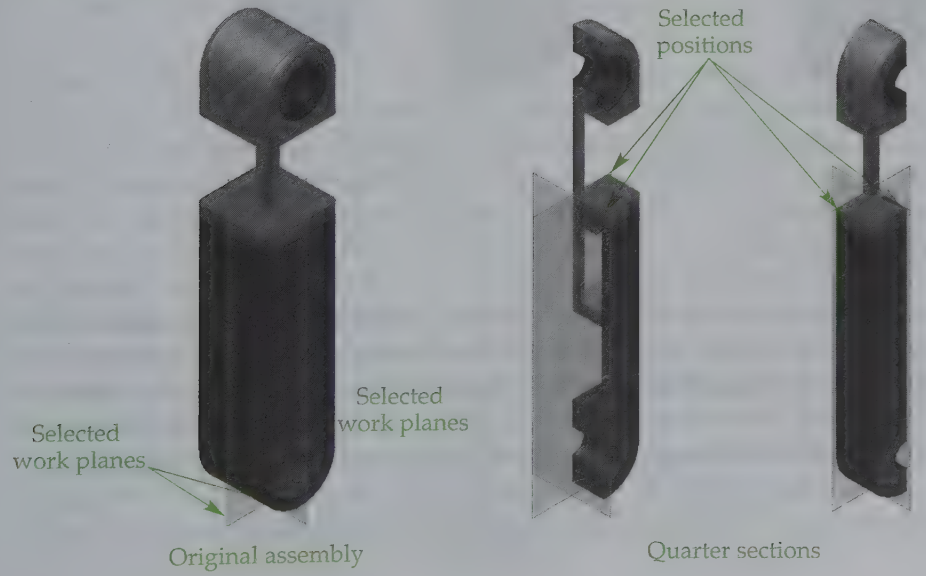


Figure 12-69.
Creating a half section view.

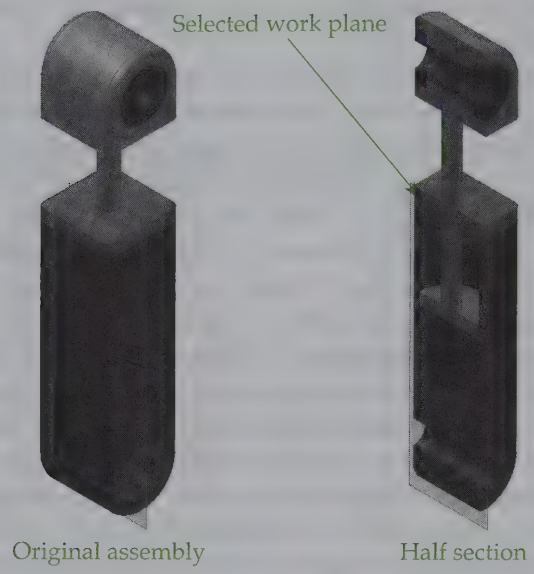
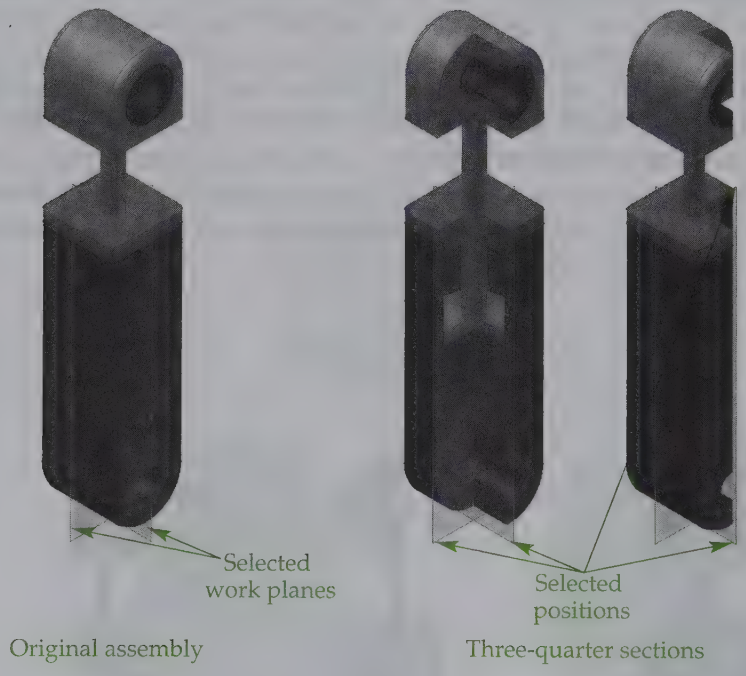
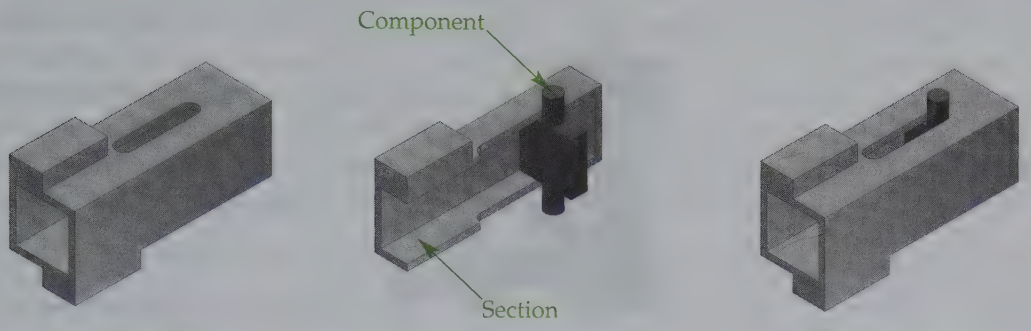


Figure 12-70.
Creating a three-quarter section view.



Once you define a section view, you can visualize the design better and observe assembly component interaction, as previously mentioned. In addition, you can sketch and create additional components in-place that would be difficult without a section. See Figure 12-71. When you finish using the section view, pick the **End Section View** button from the **Section View** flyout button to return to the complete unsectioned assembly.

Figure 12-71.
Knowing how to use section views will make it easier to create internal components in-place.



First component | Creating a component in-place | Final assembly

NOTE

Do not confuse assembly model sections with 2D drawing sections. A three-quarter assembly model section is known as a half section when drafting, and an assembly model half section is known as a full section when drafting.

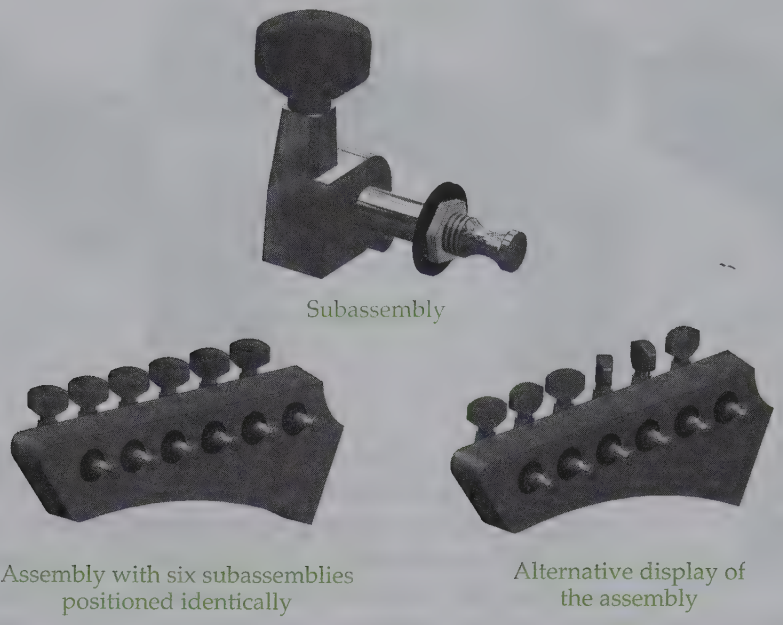
Exercise 12-23 Complete the Exercise on the Student CD.

Creating Flexible Subassemblies

Subassemblies are groups of parts that are often standard product items and can be used several times in various assemblies. A subassembly is made of the same components and looks the same regardless of the parent assembly to which it is added. This becomes an issue when dealing with subassemblies that must change shape or are shown differently in multiple assemblies or several times in the same assembly. For example, when the subassembly shown in Figure 12-72 is added to an assembly, each instance of the subassembly looks exactly the same, and cannot be modified when initially inserted and constrained. The entire subassembly is essentially grounded in the orientation specified in the actual subassembly file. Inventor allows you to create a ***flexible subassembly*** in order to display subassembly components in different orientations. See the alternative display in Figure 12-72 for an example of how subassemblies can be made flexible and then manipulated to display different display or kinematic conditions. The example in Figure 12-72 shows how the adjustment knob component of a subassembly can be turned for several instances in the same parent assembly file. However, flexible subassemblies are useful anytime subassembly components must be reoriented, including the display of a single subassembly that is used in multiple different assemblies.

To make a subassembly flexible, right-click on the subassembly name in the browser or the actual subassembly in the graphics window and select the **Flexible** pop-up menu option. Once flexible, unconstrained subassembly components are free to

Figure 12-72.
Multiple instances of a subassembly are initially positioned identically, but you can change the position of each to show an interesting alternative display or convey kinetic conditions. (Model courtesy of Ethan Collins)



move according to the possible degrees of freedom, and can be constrained as needed. A subassembly can be returned to its original orientation, as defined in the subassembly file, by deselecting the **Flexible** pop-up menu option.

NOTE



Parts, parent assemblies, component patterns, and weldments cannot be made flexible.

Using Design View Representations

A ***design view representation*** is a type of visual assembly copy that establishes a predefined assembly display. A design view representation takes a "picture" of an assembly complete with the specified view position, such as isometric, the defined set of component colors and styles, zoom, component selection, visibility status and flexibility condition. Design view representations can potentially have multiple uses because of the variety of assembly content they store. For example, you can create design view representations with different colored components, with a section view of the assembly, or with visibility of certain components turned off.

All design view representations are contained in a separate design view .idv file, automatically named the same as the assembly file. Each individual design view representation does not require its own file. Design view representation files contain only display information. You may have already noticed a default design view representation is generated when you save an assembly file. This design view representation is used to save and display assembly specifications when you reopen an assembly.

To create your own design view representation, first identify how you want

the view to look by defining the colors, visible components, or the design view representation purpose. Then use one of the following options to access the **Design View Representations** dialog box, shown in Figure 12-73, pick the **Design Views Representations** button on the browser; pick the **Design View Representations** flyout button on the browser, and select the **Other...** option; select the **Design View Representations...** option of the **View** pull-down menu; double-click on the **Private...** option in the **View** node of the **Representations** folder in the browser; or right-click on the **Private...** option in the **View** node, or the **Views** node itself, of the **Representations** folder in the browser, and select the **Edit** pop-up menu option.

Figure 12-73.
The **Design Views** dialog box.



Typically, it is appropriate to store design view representations in the assembly file in which you are currently working. This is accomplished by selecting the **Public** radio button in the **Storage Location** area of the **Design View Representations** dialog box. However, you can specify a different storage location by picking the **Private File** radio button, followed by selecting the **Browse** button and locating another existing .idv file. You may choose to rename the selected .idv file if necessary. To create a new design view representation .idv file, based on the current display characteristics, specify a storage location path and file name in the **Storage Location Path** edit box, then enter a new design view representation name in the edit box of the **Design View Representation** area, and pick the **New** button. The new design view representation should be displayed in the list box.

NOTE



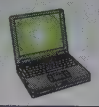
You can also create a new design view representation by right-clicking on the **View** node of the **Representations** folder in the browser and selecting the **New** pop-up menu option.

You can toggle through any existing design view representation while still using the **Design View Representations** dialog box by picking the design view representation from the list and selecting the **Active** button. The active design view displays the view characteristics you predefined before the design view representation was created. Design view representations can also be activated using one of these options: select a design view representation from the **Design View Representations** flyout button on

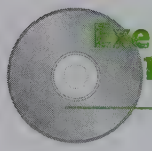
the browser; double-click on a design view representation in the **View** node of the **Representations** folder in the browser; or right-click on a design view representation in the **View** node of the **Representations** folder in the browser, and select the **Activate** pop-up menu option.

You can delete design view representations while still using the **Design View Representations** dialog box by selecting the design view representation you want to remove and choosing the **Delete** button. Design view representations can also be removed by right-clicking on a design view representation in the **View** node of the **Representations** folder in the browser, and select the **Delete** pop-up menu option.

PROFESSIONAL TIP



You may want to assign design view representation names such as half section or bracket-off, to relieve any design view confusion.



Exercise 12-24 Complete the Exercise on the Student CD.

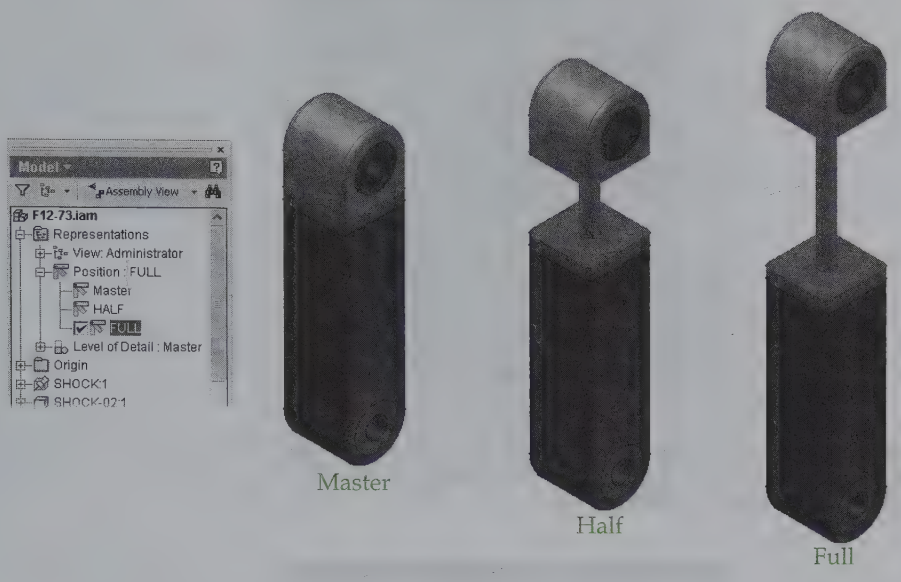
Using Positional Representations

A ***positional representation*** is a type of visual assembly copy that establishes a predefined assembly display based on the desired position of assembly components. The main purpose of a positional representation is to provide a work environment that can be used to override specific assembly constraints, without modifying or eliminating the original assembly constraints. Positional representations are also required for generating overlay drawing views in the assembly drawing environment. A basic example of using positional representations is shown in Figure 12-74. Shown is a fully constrained, nonextended assembly named Master that contains a contacting mate constraint with 0 offset, a positional representation named HALF with a mate constraint override of 1.625", and a positional representation named FULL with a mate constraint override of 3.25".

Positional representations are located, created, and modified in the browser. By default, Master is the only positional view available and defines the current work environment. As the name implies, the master positional representation identifies the primary or unmodified assembly. The master positional representation in Figure 12-74 is the fully constrained nonextended assembly. The master positional representation is the default assembly display that is used to create and manipulate assembly models. All of the assemblies you have produced so far have been associated with the master positional representation. The master positional representation only becomes relevant when you create additional positional representations.

To create a positional representation, right-click on the **Position** option of the **Representations** folder in the browser and select the **New** pop-up menu option. Most assembly tools are no longer available to use and a new positional representation named PositionalRep1 is displayed, in addition to the master positional representation, in the **Representations** folder in the browser. You can change the default name of the new positional representation by picking the name, then picking the name again, followed by entering a new name, and picking outside of the edit box. Several tools, such as **Move**

Figure 12-74.
Master is the default positional representation, but others can be created, such as the HALF and FULL positional representations that were created to illustrate various piston extensions.



Component, **Rotate Component**, and the various section view options can be used when creating a positional representation. However, the main purpose of establishing a positional representation is to override existing component placement and constraints. This process is similar to dragging, modifying, or driving constraints and components, but is accomplished using the **Override Object** dialog box. See Figure 12-74. The options available in, and the way you access the **Override Object** dialog box are specific to the item you override. For example, if you double-click or right-click on a constraint and select the **Override** pop-up menu option, the **Constraint** tab of the **Override Object** dialog box is applicable. Or, if you right-click on a component pattern and pick the **Override** pop-up menu option, the **Pattern** tab of the **Override Object** dialog box is applicable. The following information describes each of the items found in the **Override Object** dialog box:

- **Positional Representation.** This drop-down list contains options that allow you to choose which positional representation in which you want the override to occur. If you pick the **All** option, the override is applied to all positional representations except for master.
- **Constraint.** The options in this tab, shown in Figure 12-75, are available when you override a constraint. Use the **Suppression** and **Value** areas to override the suppression and value settings of the selected constraint. To override the suppression of an existing constraint, pick the **Override** check box in the **Suppression** area, and then select **Suppress** to suppress an enabled constraint, or **Enable** to enable a suppressed constraint. In order to override the value of a constraint, pick the **Override** check box in the **Value** area, and then enter or select a different value using the edit box.
- **Pattern.** The options in this tab, shown in Figure 12-76, are only available when you override a rectangular or circular component pattern. If you selected a rectangular pattern you can choose to override the row spacing, the column spacing, or both the row and column spacing by picking the appropriate **Override** check boxes. Then enter or select a different value using the corresponding edit boxes. The values you choose override the existing distance between the rectangular pattern copies, even though the value is identified as offset. If you selected a circular pattern you can choose to override the pattern angle by

Figure 12-75.
The **Override Object** dialog box, **Constraint** tab.

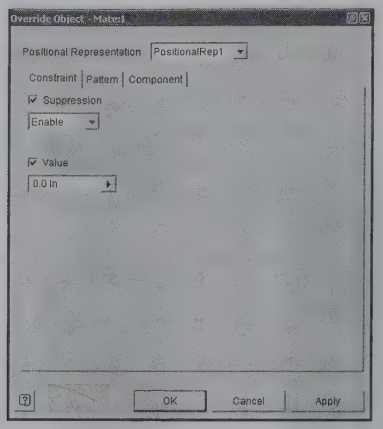
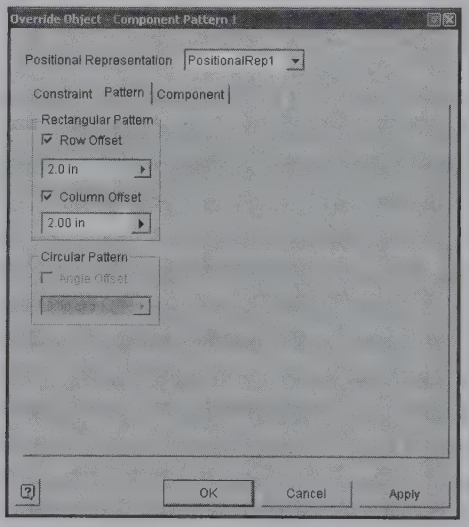


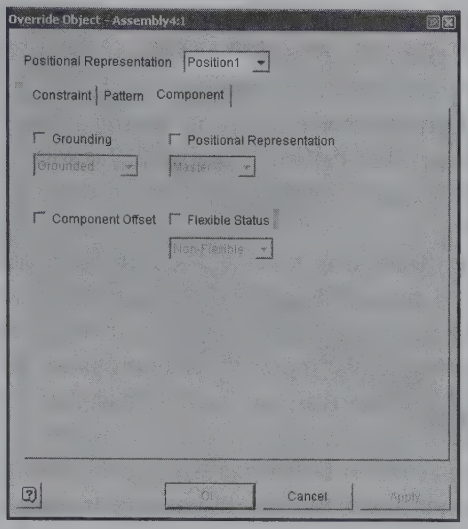
Figure 12-76.
The **Override Object** dialog box, **Pattern** tab.



picking the **Override** check box. Then enter or select a different value using the edit box. The value you choose overrides the existing spacing between circular pattern copies, even though the value is specified as offset.

- **Component.** The options in this tab, shown in Figure 12-77, are available when you override a component. Use the **Grounding** and **Offset** areas to override the grounding and offset settings of the selected component. To override the grounding of an existing component, pick the **Override** check box in the **Grounding** area, and then select **Grounded** to ground an ungrounded constraint, or **Ungrounded** to unground a grounded component. In order to override the location, or offset of a component, pick the **Override** check box in the **Offset** area, and then drag the component to a new location in the graphics window. The display boxes in the **Offset** area are for reference only and cannot be used to modify values. Pick the **Positional Representation** check box to select a positional representation from the drop-down list and override the current positional representation. If the selected component is a subassembly, pick the **Flexible Status** check box to choose either **Flexible** or **Non-Flexible** from the drop-down list and override the flexibility of the subassembly.

Figure 12-77.
The **Override Object** dialog box, **Component** tab.



NOTE

If you fully understand how to edit components, and drive constraints, as described earlier in this chapter, you are better prepared to add positional representations.

Positional representations can be activated using one of these options: double-click on a positional representation in the **Position** node of the **Representations** folder in the browser; or right-click on a positional representation in the **Position** node of the **Representations** folder in the browser, and select the **Activate** pop-up menu option. You can delete design positional representations by right-clicking on a design view representation in the **Position** node of the **Representations** folder in the browser, and selecting the **Delete** pop-up menu option.

PROFESSIONAL TIP

The master positional representation cannot be modified or removed. Use the master positional representation to return to the standard assembly work environment, as a point of beginning when creating new positional representations, and as the default positional representation view.

Exercise 12-25 Complete the Exercise on the Student CD.

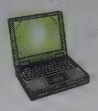
Managing Large Assemblies

Very large assemblies with many components can often be cumbersome to work with as large assembly files reduce performance and consume significant memory. One option for reducing performance problems associated with large assemblies is to postpone updates of changes made to components until you pick the **Update** button on the **Standard** toolbar. This option can be set by selecting **Defer Updates** from the **Tools** pull-down menu or by selecting the **Defer Updates** check box of the options dialog box, **Assembly** tab.

Another technique for increasing large assembly performance is to suppress components. The number of components suppressed in the assembly can be controlled using the **Level of Detail** browser options located in the **Representations** folder. By default you can select Master, All Components Suppressed, All Parts Suppressed, or All Content Center Suppressed, depending on which assembly items you want to display. To suppress all components in the assembly, double-click on the All Components Suppressed option. Use this option before saving and closing the file to increase load time when the file is opened, or to suppress all components and then individually unsuppress certain components. Double-click the All Parts Suppressed level of detail option to increase large assembly performance and work only with subassemblies. Double-click the All Content Center Suppressed level of detail option to increase large assembly performance by suppressing all content center components.

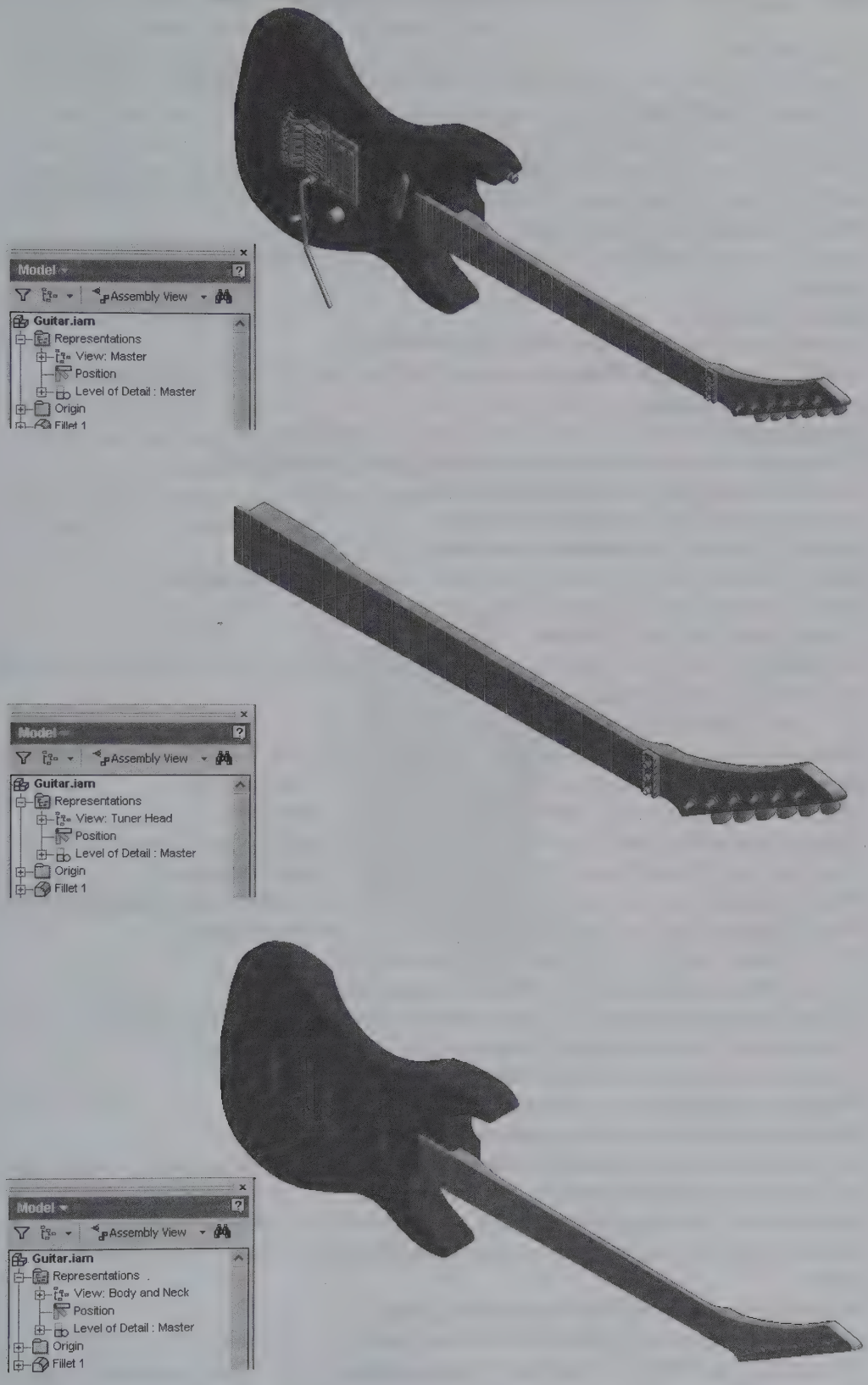
To create a custom level of detail, right-click on the **Level of Detail** option of the **Representations** folder in the browser and select the **New Level of Detail** pop-up menu option. You can change the default name of the new level of detail by picking the name, then picking the name again, followed by entering a new name, and picking outside of the edit box. Once the new level of detail is created, suppress the assembly components you do not want to display. Then, save the file to save the changes made to the level of detail. You can now work between different levels of detail to suppress and unsuppress components. See Figure 12-78.

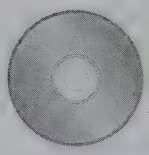
PROFESSIONAL TIP



The master level of detail cannot be modified or removed. Use the master level of detail to return to the standard assembly work environment, as a point of beginning when creating new detail levels, and as the default assembly view.

Figure 12-78.
An example of a guitar assembly with three detail levels used for working on specific portions of the assembly and increasing model performance.





Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. Define *assembly.*
2. How are multiple parts and subassemblies brought together to form an assembly in an Inventor assembly file?
3. How are constraints used in the assembly environment?
4. Identify the Inventor assembly file extension.
5. Briefly explain the two basic options for developing assemblies and subassemblies.
6. Where are assembly tools accessed?
7. Identify the first component that you should typically insert when creating an assembly.
8. Define *grounded.*
9. Give the basic function of the **Place Component** tool.
10. In addition to using the **Place Component** tool, list two techniques for placing assembly components.
11. Provide another name for *shared content* and discuss its use.
12. What does it mean to create components in-place in assemblies?
13. Discuss the function of **assembly constraints**.
14. Explain the function of **motion constraints**.
15. Give the function of **transitional constraints**.
16. Provide another name for *dragging constraints* and specify its function.
17. Discuss what iMates allow you to do.
18. Give the basic function of **iMates**.
19. Give the name of two or more iMates in a single component that are used to constrain the same component together.
20. List at least two uses for driving a constraint.
21. Name the Inventor tool that allows you to quickly generate a circular or rectangular pattern from a single component or group of existing components.
22. Describe and give a use for **mirrored components** in assemblies.
23. Explain the method used to quickly create an exact copy of parts or subassemblies, which can then be modified without affecting the original components.
24. Discuss and give an example of using the **Replace Component** and **Replace All** tools.
25. What happens when you move a component that is *not* constrained?
26. Give the basic function of the **Move Component** tool.
27. Explain the basic function of the **Rotate Component** tool.
28. Why is it sometimes a good idea to change component colors in assemblies?
29. Give a reason for sectioning an assembly.
30. Describe the effect of a **quarter section**.
31. What is the appearance of a **half section**?
32. Briefly discuss the results of a **three-quarter section**.
33. Explain the function of a **design view representation** and give an example of its use.
34. Define **flexible subassembly** and give an example.
35. Explain why it can be a problem when a subassembly is made of the same components and looks the same regardless of the parent assembly to which it is added.
36. Where are design view representations found?
37. What are design view representations automatically named?
38. Discuss the function of a positional representation and give an example of its use.
39. Where are positional representations located, created, and modified?
40. Describe the default positional view.

Problems

Instructions:

- Open the specified existing assembly file, or create new assemblies as needed.
- Insert the required parts or subassemblies.
- Create parts or subassemblies in-place as needed.
- Apply the required constraints.
- When sketches, dimensions, and other component and assembly geometry are not provided, use specifications of your choice.
- Do not change the component material or color styles unless otherwise indicated.

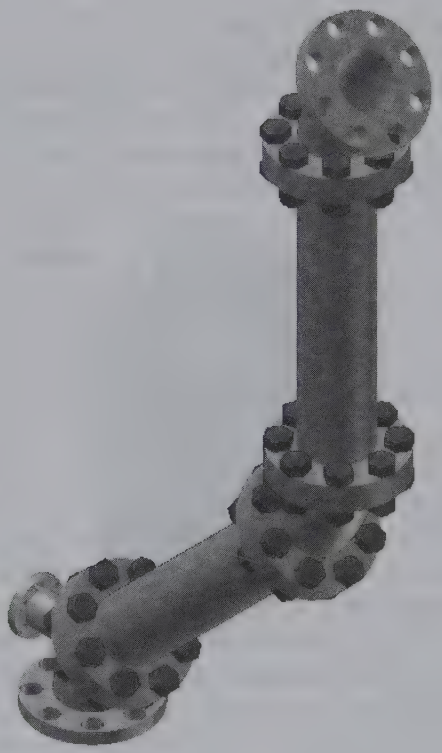
1. **Name:** Bottle Assembly
   Units: Inch
   Template: ASSEMBLY-in.iam
   Specific Instructions:
   - Insert P6-8.ipt.
   - Insert P7-8.ipt.
   - Constrain the assembly using an assembly mate constraint.
   - If you did not change the part material or color styles in the part files, change the color of each part in the assembly.

   The final assembly should look similar to the assembly shown.
   Save As: P12-1.iam



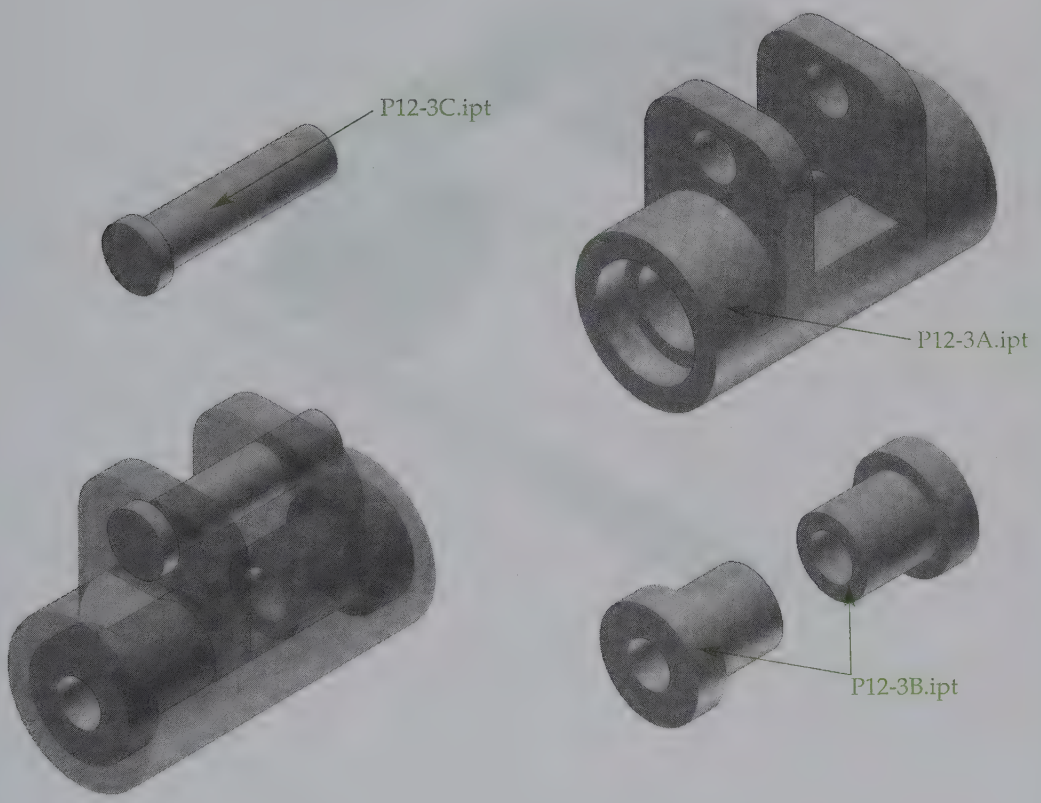
2. **Name:** Pipe Assembly
Units: Metric
Template: ASSEMBLY-mm.iam
Constraints Used: Assembly insert, flush, and mate
Pipe Color: Metal-Steel
Specific Instructions:
   - Insert two copies of P7-9.ipt.
   - Insert two copies of P8-2.ipt.
   - Create the 300 mm total length pipe shown in-place. Name the file P12-2.ipt.
   - Create a hex bolt and nut in-place. Name the bolt file P12-2_BOLT.ipt and the nut file P12-2_NUT.ipt.
   - Constrain the assembly using assembly flush and mate constraints.
   - Apply a Steel, Mild material and an As Material color to the pipe, bolt, and nut component files.

   The final assembly should look similar to the assembly shown.
   Save As: P12-2.iam



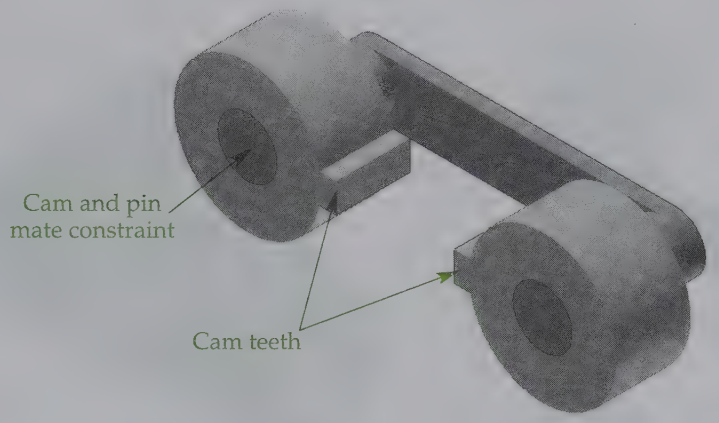
3. **Name:** Rotating Hinge Assembly
Units: Inch or Metric
Template: ASSEMBLY-in.iam or ASSEMBLY-mm.iam
Specific Instructions:
- Create the housing shown inside the assembly file. Name the part P12-3a.ipt. Use dimensions of your choice.
- Create the bushing shown inside the assembly file using the housing for reference (adaptive). Name the part P12-3B.ipt. Use dimensions of your choice.
- Build the pin shown inside the assembly file. Name the part P12-3C.ipt. Use dimensions of your choice.
- Insert another P12-3B.ipt.
- Constrain the assembly using assembly insert constraints.
- Accept the Default material and color styles for each part, and change the color of the housing to Blue (Clear) for clarity, the bushing to Yellow (Flat), and the pin to Beige (Light) inside the assembly file.

The final assembly should look similar to the assembly shown.
Save As: P12-3.iam



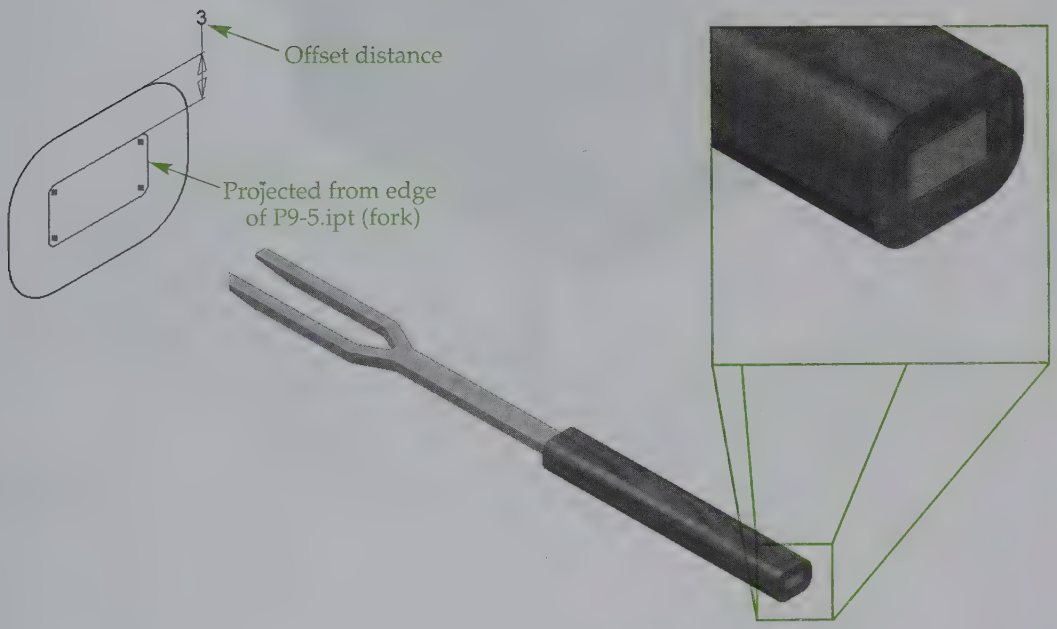
4. **Name:** Cam Assembly
Units: Inch
Specific Instructions:
- Save a copy of EX12-10.iam as P12-4.iam.
- Close EX12-10.iam without saving.
- Open P12-4.iam.
- Remove adaptivity of EX12-10b.ipt by right-clicking on EX12-10b.ipt in the browser and deselecting the **Adaptive** pop-up menu option.
- Remove the mate constraint between the bracket axis and cam axis.
- Suppress the flush constraint between the two cam teeth.
- Drive the angle constraint to observe cam rotation.
- If you did not change the part material or color styles in the part files, change the color of the cam to Red (Flat) and the color of the bracket to Aluminum (Flat).

The final assembly should look similar to the assembly shown.
Save As: P12-4.iam



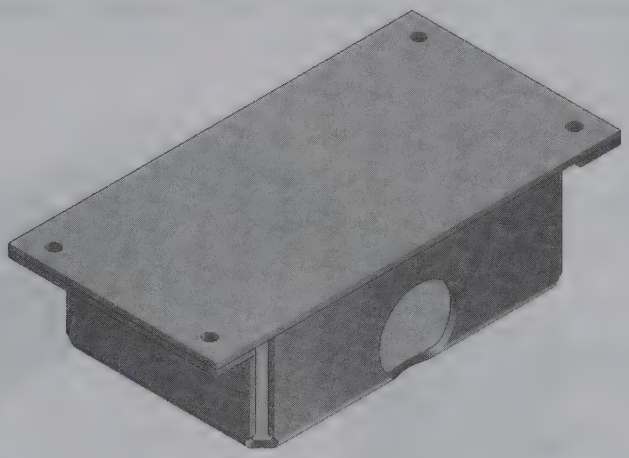
5. **Name:** Fork
Units: Metric
Template: ASSEMBLY-mm.iam
Specific Instructions:
- Insert P9-5.ipt.
- Create the fork handle shown inside the assembly file using the fork for reference (adaptive). To build the fork handle:
  1. Place a new part on the end face of the fork as shown. Name the part P12-5.ipt
  2. Open a sketch on the end face of the fork as shown, and project the fork edges.
  3. Offset the projected fork loop 3 mm away from the fork, and extrude the handle profile 90 mm along the fork.
  4. Apply a Default material, and a Wood (Ash) color to the handle part file.
- The final assembly should look similar to the assembly shown.

Save As: P12-5.iam



6. **Name:** Housing Assembly
Units: Inch
Template: ASSEMBLY-in.iam
Specific Instructions:
   - Insert P11-2.ipt.
   - Add the housing cover as shown using projected edges for reference. Be sure you use the SHEET METAL-in.ipt template. Name the new part P12-6.ipt.
   - Constrain the assembly using assembly flush, and mate constraints.
   - If you did not change the part material or color styles in the part files, specify a Default or custom material and color in both component files.

   The final assembly should look similar to the assembly shown.
Save As: P12-6.iam

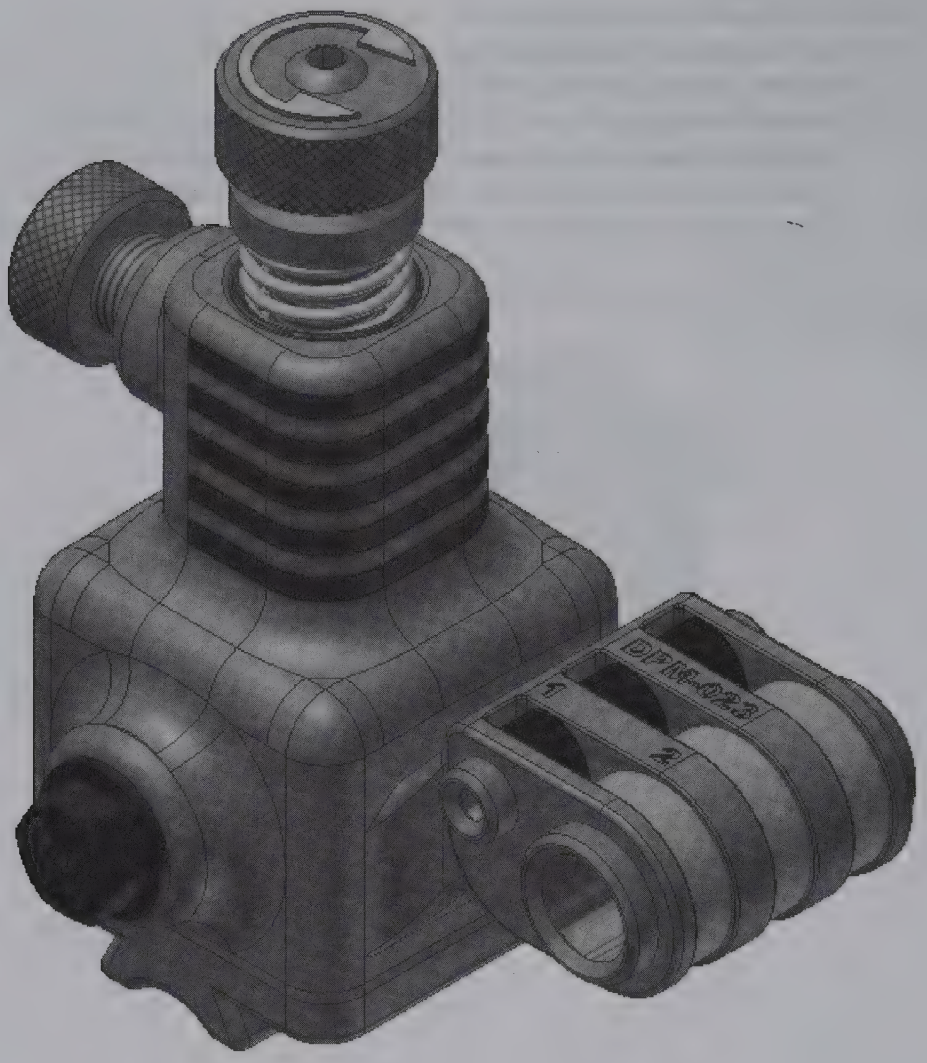


7. **Name:** Push Pin
Units: Inch
Template: ASSEMBLY-in.iam
Specific Instructions:
- Insert P5-12.ipt.
- Add the pin using the following instructions:
  - Use the **Create Component** tool to add a new part named P12-7.ipt. Open a sketch on the bottom of the head as shown.
  - Project the edge of the hole if it is not automatically projected.
  - Finish the sketch and extrude the profile .75″ midplane.
  - Chamfer the point of the pin using a .15″ × .02″ chamfer.
  - Change the pin material to Steel, High Strength Low Alloy and the color to Chrome.

The final assembly should look like the assembly shown.
Save As: P12-7.iam



All successful assemblies, complex and simple, are constructed using proper mating constraints. The components may be built outside or within the assembly. This valve assembly has mate, angle, tangent, and insert constraints to locate the components.



CHAPTER 13

Weldments

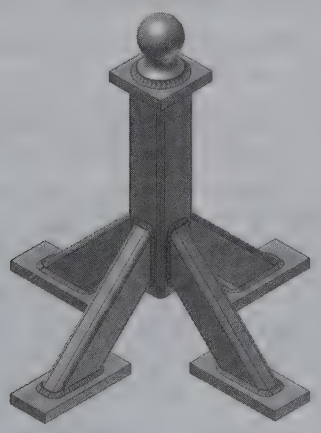
Learning Objectives

After completing this chapter, you will be able to do the following:

- Prepare weldment components for welding.
- Place fillet, cosmetic, and groove welds.
- Add welding symbols and end fills.
- Machine weldments.
- Convert an assembly to a weldment.
- Edit weldments.

Welding is a process of joining two or more pieces of like materials by heating the material to a temperature that is high enough to cause softening or melting, so that the pieces combine grain structure and become joined together. Most welding operations are performed by filling a heated joint between pieces of metal or plastic with molten material. The term ***weldment*** is used to describe components that have been welded together to form a fixed assembly. See Figure 13-1.

Figure 13-1.
An example of an Inventor weldment.



Developing Weldment Models

A weldment is a type of assembly that contains additional preparation, weld, and machined features. Just as with an assembly, a weldment consists of one or more design components, including parts, subassemblies, and subassembly weldments. In an assembly weldment file, multiple parts, subassemblies, and weldments are brought together, and constrained to form a welded assembly. As with an assembly, you can insert existing components, create components in-place, or use a combination of assembly creation techniques to form the basic weldment structure. Once the initial constrained assembly is created, there are three stages to weldment development using Inventor. The stages are material preparation, weld placement, and machining. The following information describes the typical steps required for developing a weldment:

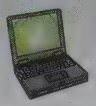
1. Open a weldment template file.
2. Insert components and create components in-place.
3. Constrain components as required to form a complete assembly.
4. Add weld preparations if needed, using preparation features.
5. Weld components together using weld features.
6. Machine the weldment, if needed, using machined features.

Weldment specific tools and options are used to add preparation, weld, and machined features to the basic assembly in order to produce a weldment. Do not confuse preparation, weld, and machined features with those used to create part or subassembly models, even though they function exactly the same. Using these tools to modify assemblies and create weldments does not affect individual part or subassembly models outside of the parent weldment file.

Typically, when you begin a new design and open a new assembly file, you specify whether the model is for an assembly application or a weldment application, depending on the type of assembly you want to create. Assembly files and weldment files carry the same .iam file extension. However, specific weldment templates are available in order to generate weldments. By default, weldment templates are listed as Weldment.iam, and are similar to other assembly templates. However, opening a new Weldment.iam file from the **New File** menu creates an assembly as a weldment subfile, which contains the weldment tools and options necessary to create weldments.

When beginning a new weldment design, you should use a weldment template. However, if you begin a weldment in an assembly application and want to access weldment tools, pick the **Weldment** option from the **Applications** pull-down menu. This menu option allows you to change from an **Assembly** design to a **Weldment** design while in a standard assembly work environment. Once you transition from an assembly to a weldment, you will not be able to return to an assembly application, and are prompted to continue.

PROFESSIONAL TIP



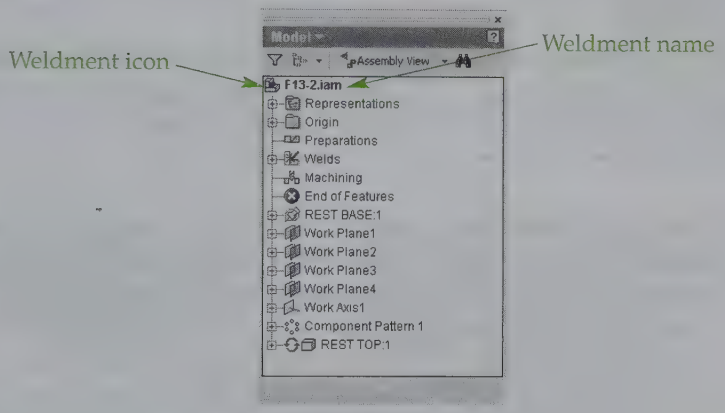
The weldment browser is slightly different than browsers in other Inventor work environments. As shown in Figure 13-2, the weldment browser contains the **Preparations**, **Welds**, and **Machining** groups, or nodes, which are similar to folders and correspond to the weldment design stages. Initially all three groups are empty, but when you prepare material, add welds, and machine the weldment, features are added. To generate weldment preparations, welds, and machining operations, you actually edit the corresponding group and enter a specific work environment that allows you to create preparation, weld, and machined features. This process is very similar to editing an existing component in an assembly file.

Exercise 13-1 Complete the Exercise on the Student CD.

Exercise 13-2 Complete the Exercise on the Student CD.

Exercise 13-3 Complete the Exercise on the Student CD.

Figure 13-2.
The browser display for a typical weldment.

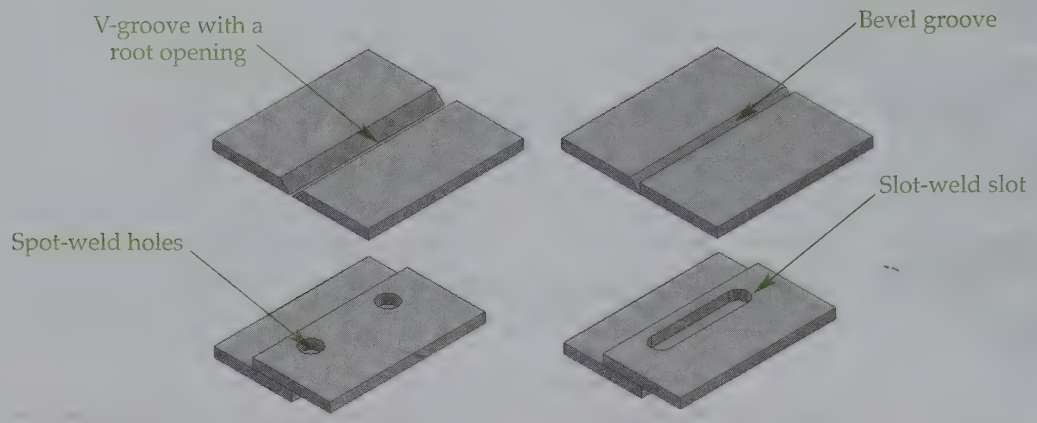


Material Preparation

Material preparation involves any task required and performed before you weld assembly components. In many situations, such as with a square groove weld, preparing the material may not be necessary. However, often a preparation feature such as a joint, groove, hole, or other feature is needed before welding. In addition, a ***root opening***, which is a specified space between components, may be required. The type, location, and size of preparation features you add to an existing assembly are specific to the welding application. Some commonly used weld preparations are shown in Figure 13-3.

Though you can prepare components for welding in individual part or subassembly files, Inventor provides tools for preparing material while working directly in a weldment file. Once you place or create weldment features and constrain the features to create an assembly, you are ready to prepare the material. To enter the preparations work environment, double-click on the **Preparations** group in the browser, or right-click on the **Preparations** group in the browser and select the **Edit** pop-up menu option.

Figure 13-3.
Examples of various weld preparations.



PROFESSIONAL TIP



In some situations you may want to constrain components after they are prepared. For example, an offset mate constraint can define a root opening.

When you activate preparations, you enter a simplified part feature environment that contains the **Extrude**, **Revolve**, **Hole**, **Sweep**, **Fillet**, **Chamfer**, **Move Face**, **Rectangular Pattern**, **Circular Pattern**, **Mirror Feature**, **Work Plane**, **Work Axis**, and **Work Point** tools. At this point you should be familiar with these tools, as they are covered in previous chapters. These weldment preparation tools are used the same as in any other model environment, with the exception of the **Extrude**, **Revolve**, and **Sweep** tools, which only allow you to apply a cut operation, and the **Fillet** and **Chamfer** tools, which only allow you to remove material.

You can access various weld preparation tools using many of the techniques described in the chapter where they apply. However, all weld preparation tools are available from the **Weldment Features** Panel bar, or the **Weldment Features** Panel toolbar as shown in Figure 13-4.

Material is removed when you prepare components for welding. As a result, Inventor provides the following preparation options:

- Cut-extrude a sketch profile using the **Extrude** tool.
- Cut-revolve a sketch profile using the **Revolve** tool.
- Add a hole using the **Hole** tool.
- Cut a sketch profile along a sketch path using the **Sweep** tool.
- Place rounds using the **Fillet** tool.
- Place chamfers using the **Chamfer** tool.
- Move faces to add a root opening using the **Move Face** tool.
- Pattern features such as holes and slots using the **Rectangular Pattern** and **Circular Pattern** tools.

Typically, sketched feature preparations are more complex than other weld preparations and may include features such as J grooves, U grooves, holes for spot welds, and slots for slot welds. See Figure 13-5. In contrast, chamfers allow you to quickly and easily produce various grooves, such as bevel and V, as shown in Figure 13-6. The **Fillet** tool can be used to create J grooves.

Figure 13-4.
The Weldment features panel in learning mode and in expert mode, and the Weldment Features Panel toolbar.

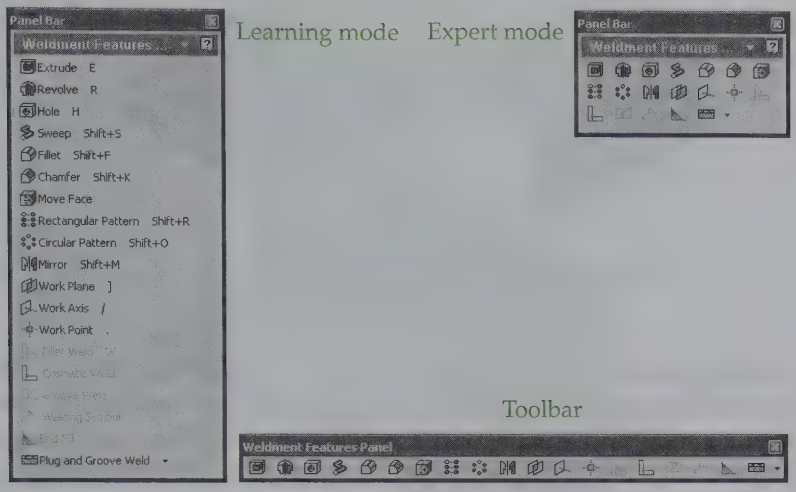


Figure 13-5.
Examples of various weld preparations created using an extruded sketch profile.

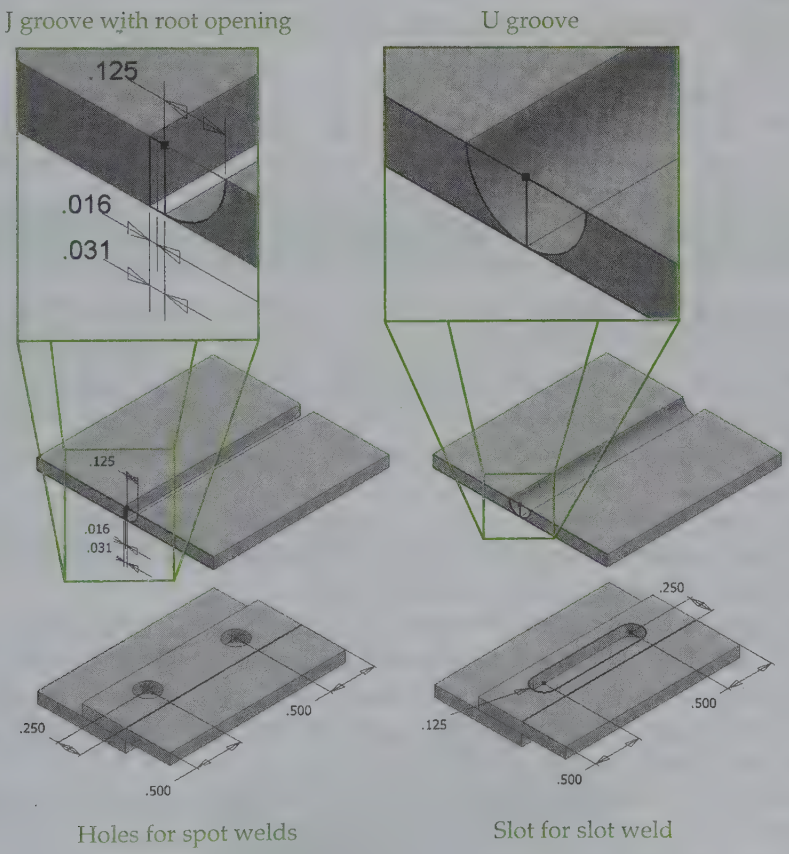
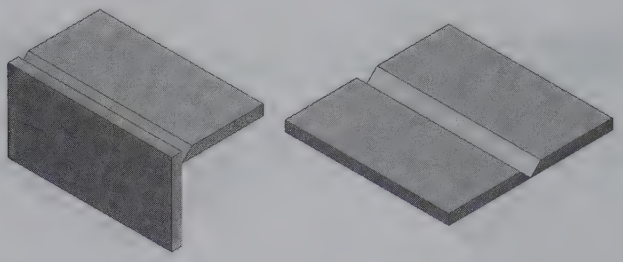


Figure 13-6.
Examples of two weld preparations created using the **Chamfer** tool.



Bevel groove V groove

Again, all material preparations are stored in the **Preparations** group in the browser. You can fully prepare a weldment by entering the preparation work environment once, or you may need to reenter **Preparations** several times throughout weldment development. When you finish preparing material, exit the preparations work environment by double-clicking on the weldment name or icon, **Welds** or **Machining** folder, or any of the weldment components in the browser; right-click and select the **Finish Edit** pop-up menu option; or pick the **Return** button on the **Inventor Standard** toolbar.



Exercise 13-4 Complete the Exercise on the Student CD.

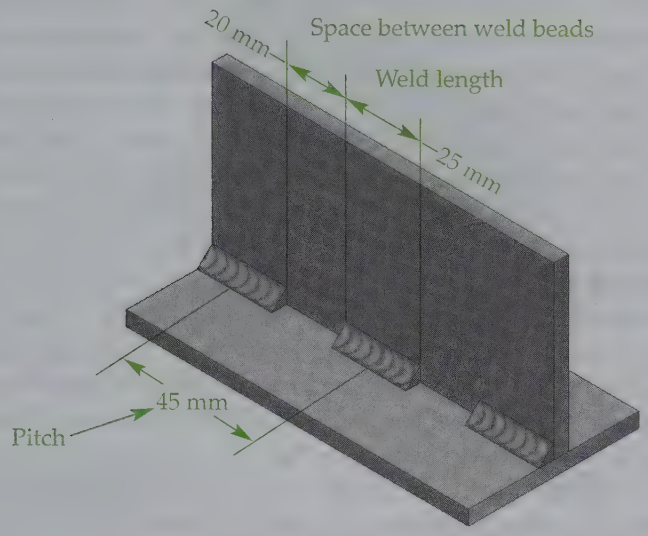
Welding

Weld placement, or ***welding***, involves specifying the type, size, and location of weld applied to assembly components. Weld placement is directly associated with material preparation, because weld material generally fills the joint, groove, hole, or other type of preparation feature, if needed. The following information provides general weld terminology, while additional terms are defined as related to specific dialog box options:

- ***Arrow side*** is the side of the weldment where the weld is selected and placed.
- ***Other side*** is the opposite side of the weldment where the weld is selected.
- ***Leg***, ***Size***, and ***Depth*** refer to the weld size.
- A weld can be placed along the entire length of a joint, or applied to a specific length and pitch. The ***weld length*** is the length of each weld, while the ***pitch*** is the center-to-center distance between weld lengths. For example, the weld shown in Figure 13-7 has a weld length of 25 mm and a pitch of 45 mm, which means there is a 20 mm space between weld beads.
- ***Contour*** is the external shape of the weld surface, such as flat, concave, or convex.

Though you can add weld information to two-dimensional drawings, Inventor provides tools for welding while in the weldment subfile, much like creating an actual weldment. Then, when you create a 2D drawing, weld information defined in the model is available to add to drawing views, in the same way that model dimensions can be displayed in a drawing. Once you place or create weldment features, constrain

Figure 13-7.
You must know the weld length and pitch to build an intermittent fillet weld.

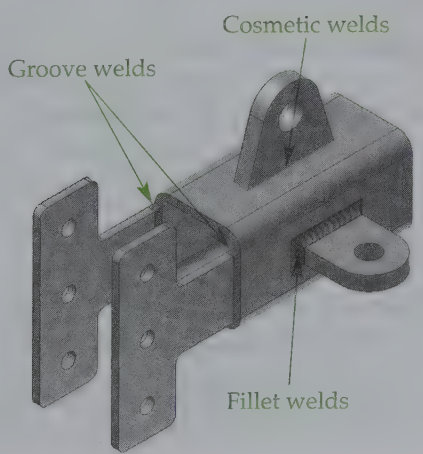


the features to create an assembly, and place any required preparation features, you are ready to weld. To enter the welding environment, double-click on the **Welds** folder in the browser, or right-click on the **Welds** folder in the browser and select the **Edit** pop-up menu option.

When you activate welds, you enter a work environment that contains the **Work Plane**, **Work Axis**, **Work Point**, **Fillet Weld**, **Cosmetic Weld**, **Groove Weld**, **Welding Symbol** and **End Fill** tools. The work feature tools are used exactly the same as in any other model environment. However, the **Fillet Weld**, **Cosmetic Weld**, **Groove Weld**, **Welding Symbol** and **End Fill** tools are specific to weldments and are used to place welds and weld information. You can access the work feature tools using many of the techniques described in the chapter where applied. However, the work feature and welding tools are all available from the **Weldment Features Panel** bar and **Weldment Features Panel** toolbar.

Inventor allows you to create fillet, cosmetic, and groove welds. See Figure 13-8. As you will discover, the process for placing a fillet, cosmetic, or groove weld is very

Figure 13-8.
Examples of Inventor fillet, cosmetic, and groove welds.



similar. The main difference between fillet, cosmetic, and groove welds involves the weld display and property characteristics. When you create a fillet or groove weld, a .bmp image of weld geometry is shown, a weld symbol can be displayed, weld symbol information is stored in a model, and weld bead mass properties are calculated. In contrast, when you create a cosmetic weld, the selected weld intersection, or joint, is highlighted, a weld symbol can be displayed, weld symbol information is stored in a model, and basic weld bead cross-sectional area properties are calculated.

The fillet weld type is used primarily to create a true fillet weld, while a groove weld type is used to place other types of welds. Cosmetic welds can be used to add any type of weld, but again do not show .bmp weld beads as with the fillet and groove weld types. Select the weld type you feel is appropriate for the specific weldment application. To create an authentic fillet weld, use the **Fillet Weld** tool. Use the **Groove Weld** tool to create other types of non-fillet welds, such as spot, slot, V-groove, and butt. Or, you may want to use a cosmetic weld type if you do not need to display actual weld geometry in the model; you are concerned about the increased file size and generation time required to produce a fillet or groove weld type; or fillet and groove welds do not work correctly, or continually contain errors, when applied to the weldment.

All welds are stored in the **Welds** group in the browser. Weld beads are located in the **Bead** folder of the **Welds** group, and you can drag **End of Features** as needed to suppress weld beads. Welding symbols are listed separately in the browser, and consume a copy of the model weld bead, though the weld bead is also displayed in the **Beads** folder. You can fully weld a weldment by entering the welding work environment once, or you may need to reopen the welding tools several times throughout weldment development. When you finish welding, exit the welding work environment by double-clicking on the weldment name or icon, the **Preparations** or **Machining** folder, or any of the weldment components in the browser; or pick the **Return** button on the **Inventor Standard** toolbar.

Creating Fillet Welds

A ***fillet weld*** creates a triangular weld bead cross section that is typically applied to the intersection of two non-parallel assembly components. See Figure 13-9. Fillet welds are created using the **Fillet Weld** tool. Though the primary function of the **Fillet Weld** tool is to generate true fillet welds, as shown in Figure 13-9, you can use the **Fillet Weld** tool to create a variety of other welds including plug, spot, and combined groove and fillet welds. To access the **Fillet Weld** tool, pick the **Fillet Weld** button on the **Weldment Features Panel** bar, pick the **Fillet Weld** button on the **Weldment Features Panel** toolbar, or press [W].

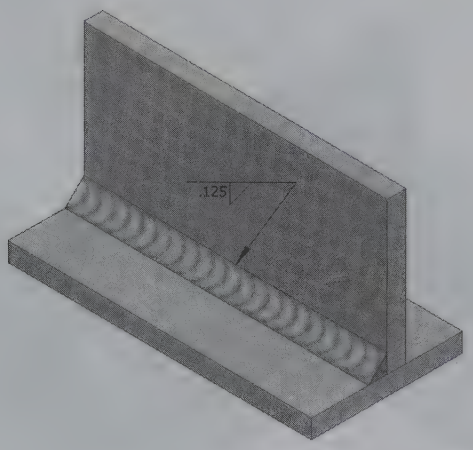
The **Fillet Weld** dialog box is displayed when you access the **Fillet Weld** tool. See Figure 13-10. The first step to creating a fillet weld is to pick the faces of the components you want to weld together. Choose the **Select Face(s) 1** button, if it is not currently active, and select the first weldment face or set of faces on which you want to apply welds. Next, pick the **Select Face(s) 2** button and choose the second weldment face or set of faces on which you want to apply welds. When selecting multiple tangent faces, it may help, and save time, to pick the **Chain** check box, so that you do not have to select the faces individually. As shown in Figure 13-11, the faces you select define the intersection of the components where the fillet weld occurs.

NOTE



To deselect faces, with the **Select Face(s) 1** or **Select Face(s) 2** still active, hold down the [Shift] or [Ctrl] key on your keyboard and pick the faces you want to remove from the selection.

Figure 13-9.
An example of a fillet weld and a weld symbol that describes the weld.



PROFESSIONAL TIP



If you are welding all-around weldment components, you must pick multiple adjacent faces for each side.

Once all faces are selected, use the following options in the **Fillet Weld** dialog box to define the size, shape, and extents of the fillet weld:

- **Leg Length Measurement.** Pick this radio button to specify the weld bead size according to the length of two legs. A weld bead *leg* is the distance from the intersection of the two selected faces, or joint, to the intersection of the weld bead and one of the component faces, or *toe*. Using the leg definition technique is much like placing a two-distance fillet or round and can be used to create a weld bead with different leg lengths. When the **Leg Length Measurement** radio button is selected, two edit boxes are available. Use the **Leg 1** edit box to define the length of the first leg, which corresponds to the first face you selected.

Figure 13-10.
The **Fillet Weld** dialog box.

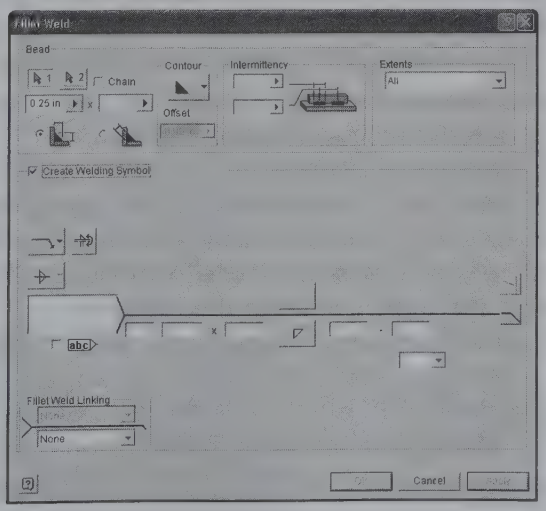
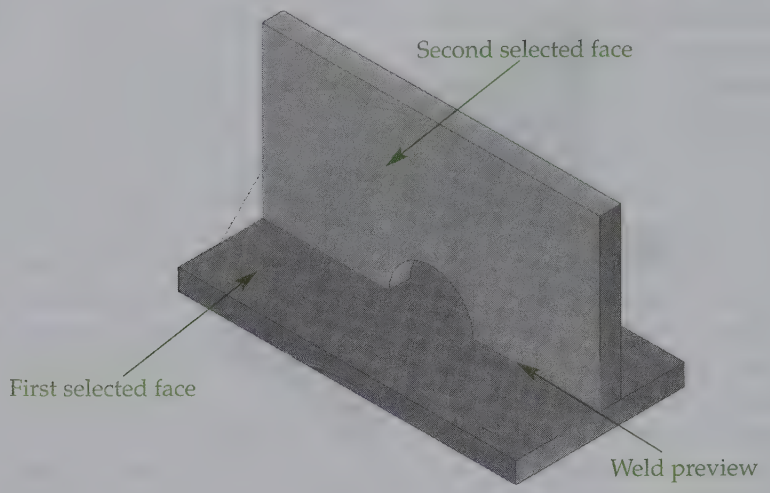


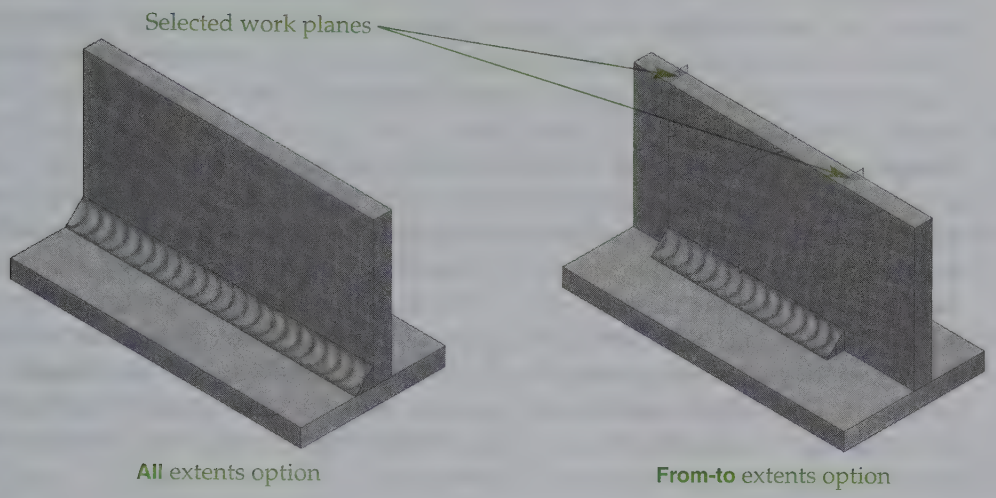
Figure 13-11.
A weld preview will appear after weld faces are selected.



Then, if necessary, use the **Leg 2** edit box to define the length of the second leg, which corresponds to the second face you selected. Notice that if both legs are the same length, you only have to enter a value in the **Leg 1** edit box.

- **Throat Measurement.** Pick this radio button to specify the weld bead size according to the weld bead *throat*, which is the shortest distance from the intersection of the two selected faces, or joint, to the weld bead face. Using throat definition technique is much like placing a single distance fillet or round and creates a weld bead with equal leg lengths. When the **Throat Measurement** radio button is selected, one edit box is available. Use the **Throat Measurement** edit box to define the depth of the weld bead throat.
- **Contour.** This area allows you to define the shape of the weld bead. Pick the **Flat**, **Convex**, or **Concave** option from the **Contour** flyout button, depending on the required weld bead contour shape. When the **Convex** or **Concave** option is selected, the **Offset** edit box becomes available allowing you to define the distance from the imaginary line connecting the fillet weld toes to the weld bead face. The larger the convex or concave offset, the smaller the radius of the weld bead contour. The **Offset** edit box does not apply to a **Flat** contour because the actual face of the weld bead connects the weld bead toes.
- **Intermittency.** Use this area to establish whether the fillet weld is placed along the entire length of a joint, or applied according to a specific length and pitch. If both edit boxes are clear, the weld bead is placed along the entire joint. To create an intermittent weld bead, enter the weld length in the **Length** edit box, and the weld pitch using the **Pitch** edit box. For more information regarding length and pitch, refer to the previous description.
- **Extents.** Use this drop-down list to define the extents of the weld. See Figure 13-12. When you pick the **All** option, a weld bead is placed along the entire selected edge. Pick the **From-To** extents option to define the extents of the weld using two parallel faces or planes. When you choose the **From-To** option, the **From** and **To** buttons are displayed allowing you to make the required selections.
- **Create Welding Symbol.** Pick this check box to add a weld symbol in addition to weld bead geometry. Welding symbols are added to a model primarily for reference purposes and help you better characterize and recognize weld beads. However, when you create a two-dimensional weldment drawing, the information you specify in the weldment model is referenced in the drawing,

Figure 13-12.
An example of a cosmetic weld created using the **All** extents option and the **From-To** extents option.



allowing you to quickly add welding symbols without recreating the weld data. When you pick the **Create Welding Symbol** check box, the **Create Welding Symbol** area is displayed. Use the following options to fully define the welding symbol:

NOTE

Many of the options in the **Create Welding Symbol** area are more appropriately used when creating cosmetic welds because cosmetic weld beads do not contain any specific model parameters like those defined when adding fillet welds.

- **Fillet Weld Linking.** This area contains two drop-down lists that allow you to define the arrow side and other side welding symbol specifications according to the weld bead settings previously described. By default the **None** option is selected, which allows you to create a welding symbol without referencing the model weld bead. If you do want to use the information specified in the **Bead** area of the **Fillet Weld** dialog box, pick the **Current Bead** option from the drop-down lists. You will notice that several of the welding symbol options become unavailable, because the data has already been set in the **Bead** area.

NOTE

The **Other Side** drop-down list of the **Fillet Weld Linking** area is only available if you specify an other side symbol using the **Other Side** button.

PROFESSIONAL TIP

Use fillet weld linking when adding a welding symbol to your fillet welds so that the symbol is associated with the model weld beads.

- **Identification Line.** This flyout button is used to control the placement of the identification line, if used. An *identification line* is placed in addition to the reference line when using British welding drafting standards. The *reference line* is the line on which weld symbol information is displayed. By default the **No Identification Line** button is selected, and an identification line is not shown. Select the **Identification Line - Arrow Side** or **Identification Line - Other Side** option from the flyout, depending on the application. The effects of the selected identification line button are displayed when the weld is created and weld symbol is displayed.
- **Swap Arrow/Other Symbols.** Use this button to define the selected side on which a weld bead is placed according to the welding symbol. For example, if you are currently defining an arrow side weld, when you pick the deselected **Swap Arrow/Other Side** button, you specify the other side instead of the arrow side.
- **Stagger.** This flyout button is only available if you specify an other-side symbol using the **Other Side** button. Use the **Stagger** flyout button to define the orientation of weld symbols. By default, the button is set to **No Stagger**, but you can choose from a variety of stagger options depending on the application and standard. Again, the effects of the selection button are displayed when the weld is created and weld symbol is shown.
- **Note.** This edit box is used to add a weld note to the symbol. To display a note, type the required information in the edit box.
- **Trail Note Box.** Depending on the specified drafting standard, choose this check box to add a box around the weld note, if you added a note to the **Note** edit box.
- **Prefix.** Use this edit box to define a weld symbol prefix. A *prefix* is a word, character, or symbol added before the weld size.
- **Leg 1.** Use this edit box to identify the length of the first weld bead leg in the welding symbol.
- **Leg 2.** Use this edit box to identify the length of the second weld bead leg in the welding symbol.
- **Arrow Side Symbol.** This button is used to describe the type of weld you create on the arrow side of the joint. If you are not placing the default fillet weld, pick the **Arrow Side Symbol** button and choose a different symbol.
- **Length.** Enter the length of an intermittent weld in this edit box.
- **Pitch.** Enter the pitch of an intermittent weld in this edit box.
- **Contour.** Select a contour from this drop-down list.
- **Other Side Symbol.** By default, an arrow side welding symbol is created, unless you pick the **Swap Arrow/Other Symbols** button previously described. In order to use a symbol with information regarding both sides, you must pick the **Other Side Symbol** button or the **Arrow Side Symbol** button if the **Swap Arrow/Other Symbols** button is active, and select a symbol from the palette. When you pick a symbol, the other side options become available. The other side preferences available are the same as the arrow side options previously described, but are located above the weld symbol reference line.
- **Field Weld.** Pick this button to add a field weld symbol for field welding applications.
- **All Around.** Select this button to add a weld all-around symbol, for applications that require all-around welding.

Once you select and enter all the required fillet weld information in using the **Fillet Weld** dialog box, you are ready to generate the weld by picking the **Apply** button. The **Fillet Weld** dialog box remains open, allowing you to place additional welds. When finished, pick the **OK** button to generate welds and exit the **Fillet Weld** tool.



Complete the Exercise on the Student CD.

Placing Cosmetic Welds

A *cosmetic weld* highlights the selected weld joint, and adds weld information to a weldment, without creating a model weld bead. See Figure 13-13. Cosmetic welds can be applied to any joint at the intersection of components, and represent a multitude of welding operations, not just fillet and groove welds. Cosmetic welds are created using the **Cosmetic Weld** tool. To access the **Cosmetic Weld** tool, pick the **Cosmetic Weld** button on the **Weldment Features Panel** bar, or pick the **Cosmetic Weld** button on the **Weldment Features Panel** toolbar.

The **Cosmetic Weld** dialog box is displayed when you access the **Cosmetic Weld** tool. See Figure 13-14. The first step to creating a cosmetic weld is to pick joints to define which components to weld together. Choose the **Select Edge(s)** button if it is not currently active, and select the intersections of weldment components at which you want to apply welds. See Figure 13-15. When selecting multiple continuous tangent edges, it may help, and save time, to pick the **Chain** radio button, so that you do not have to select the edges individually. Or, when selecting an entire closed loop, it may help, and save time, to pick the **Loop** radio button, so that you do not have to select the edges individually.

NOTE



To deselect faces, with the **Select Edge(s)** still active, hold down the [Shift] or [Ctrl] key on your keyboard and pick the faces you want to remove from the selection.

Once all joints are selected, use the following options in the **Cosmetic Weld** dialog box to define the size, shape, and extents of the cosmetic weld:

- **Extents.** Use this drop-down list to define the extents of the weld. The **Extents** area of the **Cosmetic Weld** dialog box functions the same as the **Extents** area of the **Fillet Weld** dialog box previously described.

Figure 13-13.
An example of a cosmetic weld and a weld symbol that describes the weld.

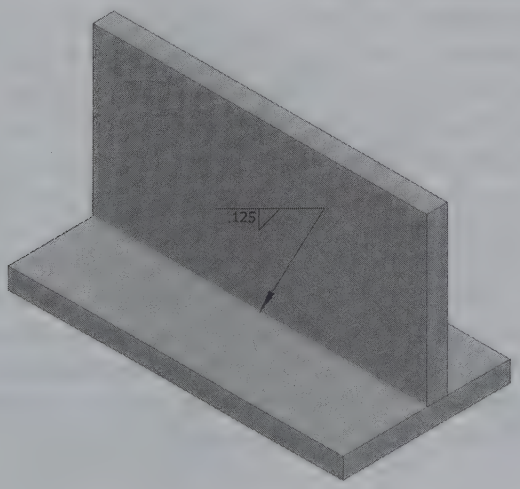


Figure 13-14.
The **Cosmetic Weld** dialog box.

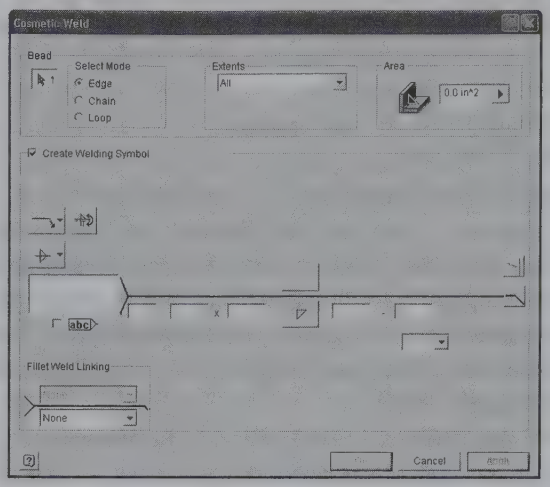
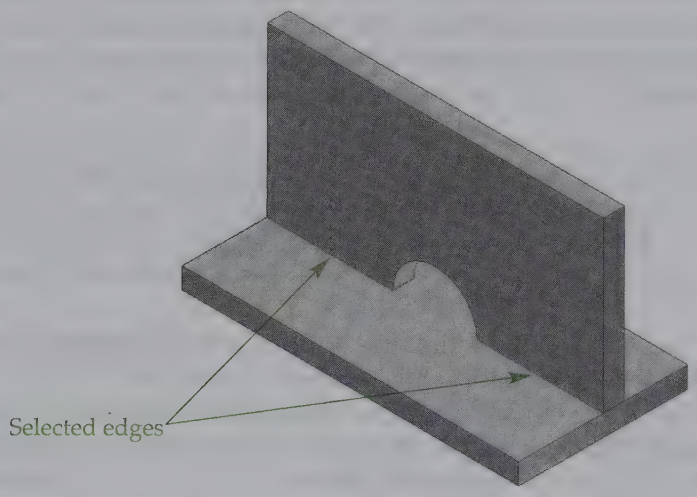


Figure 13-15.
Selecting cosmetic weld edges.



- **Area.** This area contains an edit box that allows you to define the cross-sectional area of the weld bead, even though a weld bead is not actually shown in the model. The default value is 0 to the second power, but you can enter any acceptable number or equation.
- **Create Welding Symbol.** Pick this check box to add a weld symbol in addition to weld bead geometry. Welding symbols are especially important to cosmetic welds because cosmetic weld beads do not contain any specific model parameters like those defined when adding fillet and groove welds. As a result, if you do not add a weld symbol, a cosmetic weld is a representation that implies some type of weld is added to a joint, even though cross-sectional area is calculated in the model. Additional, as with fillet welds, when you create a two-dimensional weldment drawing, the information you specify in the weldment model is referenced in the drawing, allowing you to quickly add welding symbols without recreating the weld data. When you pick the **Create Welding Symbol** check box, the **Create Welding Symbol** area is displayed. The **Create Welding Symbol** area functions the same as the **Create Welding Symbol** area of the **Fillet Weld** dialog box previously described.

Once you select and enter all the required cosmetic weld information using the **Cosmetic Weld** dialog box, you are ready to generate the weld by picking the **Apply** button. The **Cosmetic Weld** dialog box remains open, allowing you to place additional welds. When finished, pick the **OK** button to generate welds and exit the **Cosmetic Weld** tool.



Complete the Exercise on the Student CD.

Adding Groove Welds

A *groove weld* creates a weld bead that is typically applied to the groove created at the intersection of two, often parallel assembly components. See Figure 13-16. Groove welds are placed using the **Groove Weld** tool. You can use the **Groove Weld** tool to create a variety of groove welds including the square-groove weld shown in Figure 13-16, a double-V-groove, and a flare-bevel, just to name a few. The **Groove Weld** tool can also be used to add the groove weld portion of a combined groove and fillet weld. To access the **Groove Weld** tool, pick the **Groove Weld** button on the **Weldment Features Panel** bar, or pick the **Groove Weld** button on the **Weldment Features Panel** toolbar.

The **Groove Weld** dialog box is displayed when you access the **Groove Weld** tool. See Figure 13-17. The first step to creating a groove weld is to pick the faces of the components you want to weld together. Choose the **Select Face(s) 1** button, if it is not currently active, and select the first weldment face or set of faces on which you want to apply welds. Next, pick the **Select Face(s) 2** button and choose the second weldment face or set of faces on which you want to apply welds. As shown in Figure 13-18, the faces you select define the weld joint, and the groove that is filled by the weld bead.

NOTE



To deselect faces, with the **Select Face(s) 1** or **Select Face(s) 2** still active, hold down the [Shift] or [Ctrl] key on your keyboard and pick the faces you want to remove from the selection.

Figure 13-16.
An example of a groove weld.

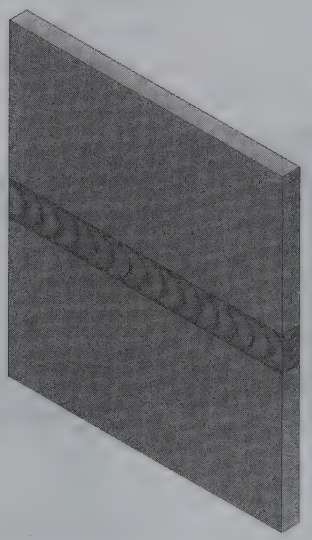
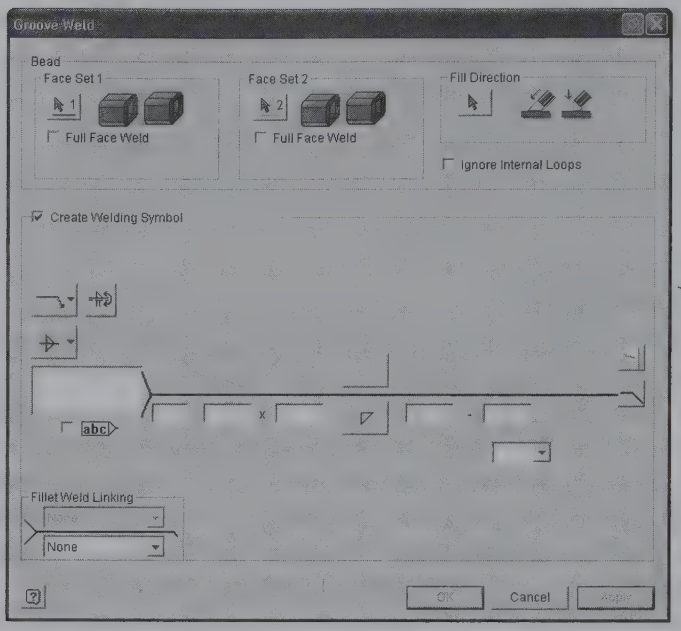


Figure 13-17.
The **Groove Weld** dialog box.



Once you choose the required faces, or anytime during the selection process, you may want to pick the following check boxes located in the **Face Set 1** and **Face Set 2** areas, depending on the application:

- **Full Face Weld.** Select this check box to define the length of the weld bead in reference to the longest face. See Figure 13-19.
- **Ignore Internal Loops.** Choose this check box to add weld bead geometry to the entire face defined by an internal loop. Without the **Ignore Internal Loops** check box selected, internal loops are calculated, and a weld bead is placed only around the groove. See Figure 13-20.

After you select the component faces and pick any required selection set options, the next step to creating a groove weld is to specify fill direction. If you pick the **Full Face Weld** check boxes in both the **Face Set 1** and **Face Set 2** areas, the fill direction is automatically calculated. However, if the **Full Face Weld** check boxes are not selected, you must choose the **Select Fill Direction** button and pick the desired fill direction. The fill direction

Figure 13-18.
Selecting groove weld faces.

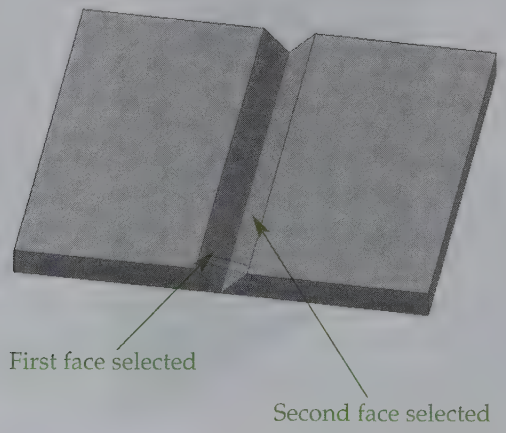
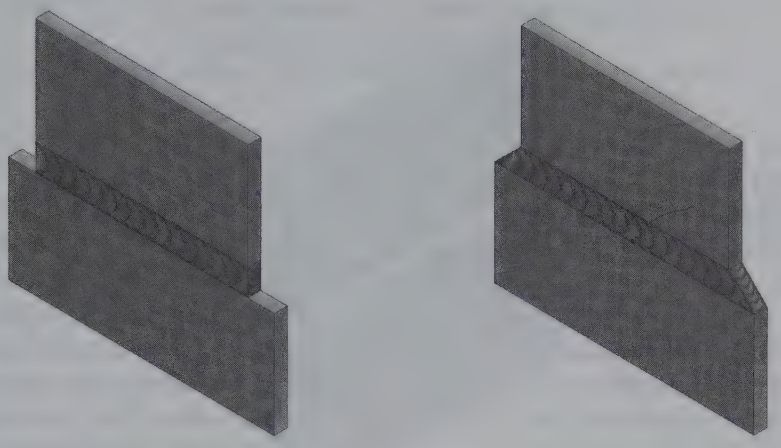


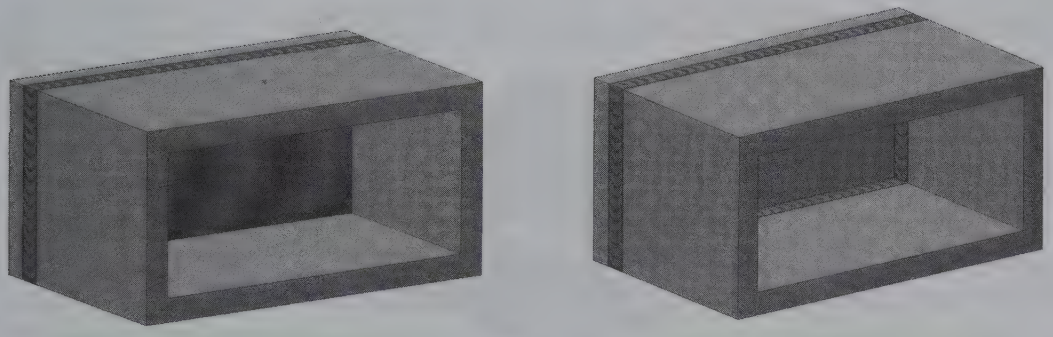
Figure 13-19.
These groove welds illustrate the effects of using the **Full Face Weld** option.



Without **Full Face Weld** option

With **Full Face Weld** option

Figure 13-20.
These groove welds illustrate the effects of using the **Ignore Internal Loops** option.

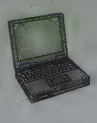


Without **Ignore Internal Loops** option

With **Ignore Internal Loops** option

describes how the weld bead is generated and the weld faces projected, in reference to the selected component faces. You can choose any appropriate face, work plane, edge, axis or two points that are associated with the selected component faces, depending on the required weld. Choosing the correct fill direction can be confusing and difficult due to the frequently large number of possible selections. Often, the fill direction should be perpendicular to the selected component faces. For example, to create the groove weld shown in Figure 13-21, the fill direction is defined by selecting the face shown, which indicates the direction as perpendicular to the groove. Figure 13-22 shows examples of a groove weld created by choosing the same face sets, but different fill directions.

PROFESSIONAL TIP



Help locate geometry to constrain using the **Select Other** tool and wheel mouse selection techniques described in previous chapters. You can cycle through possible edges, axes, planes, and other elements for constraint purposes.

Figure 13-21.
Choosing a groove weld fill direction.

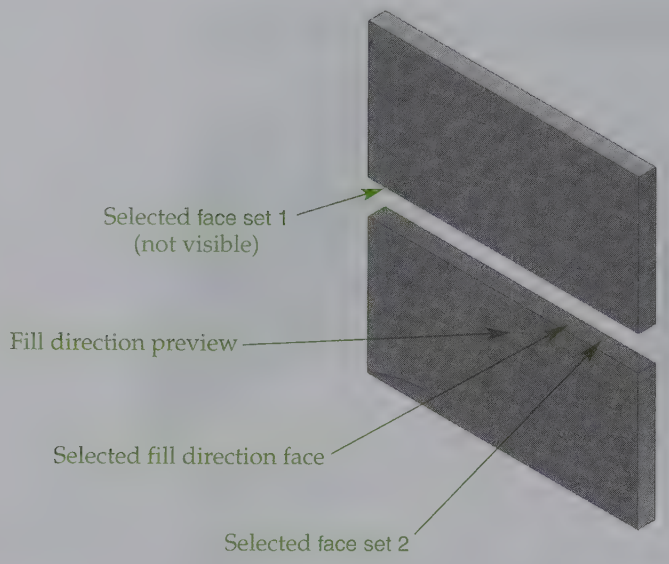
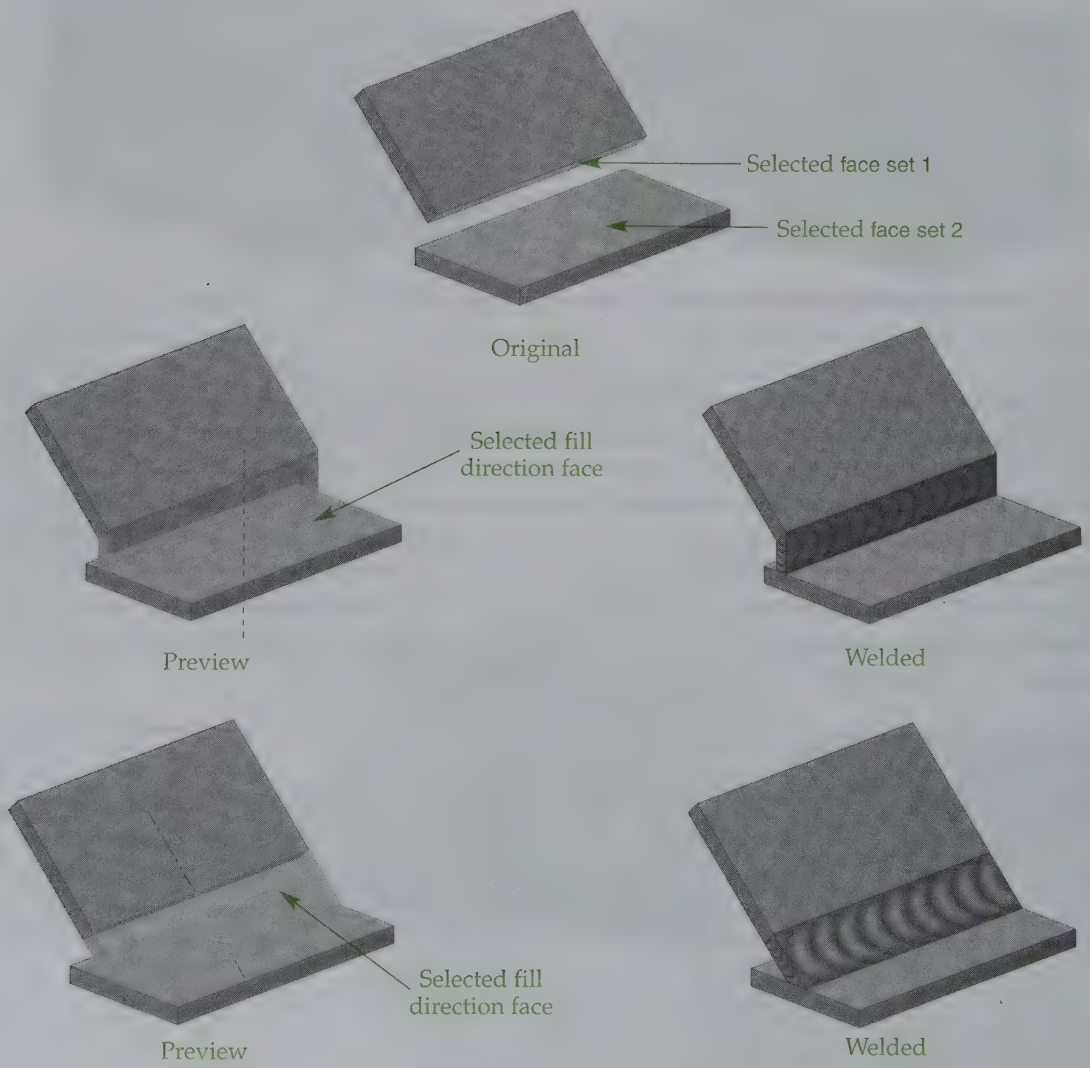


Figure 13-22.
The preview will appear after choosing a groove weld fill direction.



NOTE

Do not rely on picking both **Full Face Weld** check boxes to define the fill direction. Picking both **Full Face Weld** check boxes may result in an error that can be solved by specifying a fill direction.

Once all faces are selected and you specify a fill direction, you can add a weld symbol and further describe the groove weld by picking the **Create Welding Symbol** check box. Welding symbols are added to a model primarily for reference purposes, and help you better characterize and recognize weld beads. However, when you create a two-dimensional weldment drawing, the information you specify in the weldment model is referenced in the drawing, allowing you to quickly add welding symbols without recreating the weld data. When you pick the **Create Welding Symbol** check box the **Create Welding Symbol** area is displayed. The **Create Welding Symbol** area functions the same as the **Create Welding Symbol** area of the **Fillet Weld** and **Cosmetic Weld** dialog boxes previously described.

After you select and enter all the required groove weld information using the **Groove Weld** dialog box, you are ready to generate the weld by picking the **Apply** button. The **Groove Weld** dialog box remains open, allowing you to place additional welds. When finished, pick the **OK** button to generate welds and exit the **Groove Weld** tool.

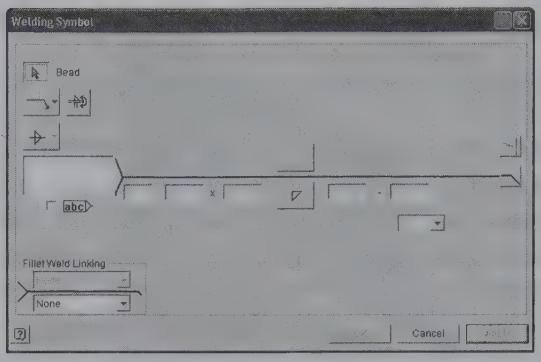
Exercise 13-7 Complete the Exercise on the Student CD.

Adding Welding Symbols

Welding symbols can be applied to fillet, cosmetic, and groove welds when you create the weld bead. This is done by picking the **Create Welding Symbol** check box and using the options in the **Create Welding Symbol** area found in each weld bead tool. The welding symbol is used in the model for reference and definition purposes, and can be referenced in the associated weldment drawing, allowing you to quickly add welding symbols without recreating the weld data. In most cases, it is easiest to add a welding symbol when you are working in the **Fillet Weld**, **Cosmetic Weld**, or **Groove Weld** dialog box. However, you can place a welding symbol without using the **Create Welding Symbol** option of these dialog boxes by accessing the **Welding Symbol** tool. Fillet, cosmetic, and groove weld features are consumed by the welding symbol if you add the welding symbol using the **Create Welding Symbol** option of a dialog box. As a result, you can only place a welding symbol using the **Welding Symbol** tool on weld beads that do not already contain welding symbol information. Assuming your model has weld beads that are not consumed by existing welding symbols, access the **Welding Symbol** tool by picking the **Welding Symbol** button on the **Weldment Features Panel** bar or picking the **Welding Symbol** button on the **Weldment Features Panel** toolbar.

The **Welding Symbol** dialog box is displayed when you access the **Welding Symbol** tool. See Figure 13-23. The first step to creating a welding symbol is to pick the **Select Bead** button, if it is not currently active, and select the weld bead on which you want to apply a welding symbol. Next, define the welding symbol using the rest of the **Welding Symbol** dialog box options. The additional options found in the **Welding Symbol** dialog box are the same as those located in the **Create Welding Symbol** area of the **Fillet Weld**, **Cosmetic Weld**, or **Groove Weld** dialog boxes, previously described.

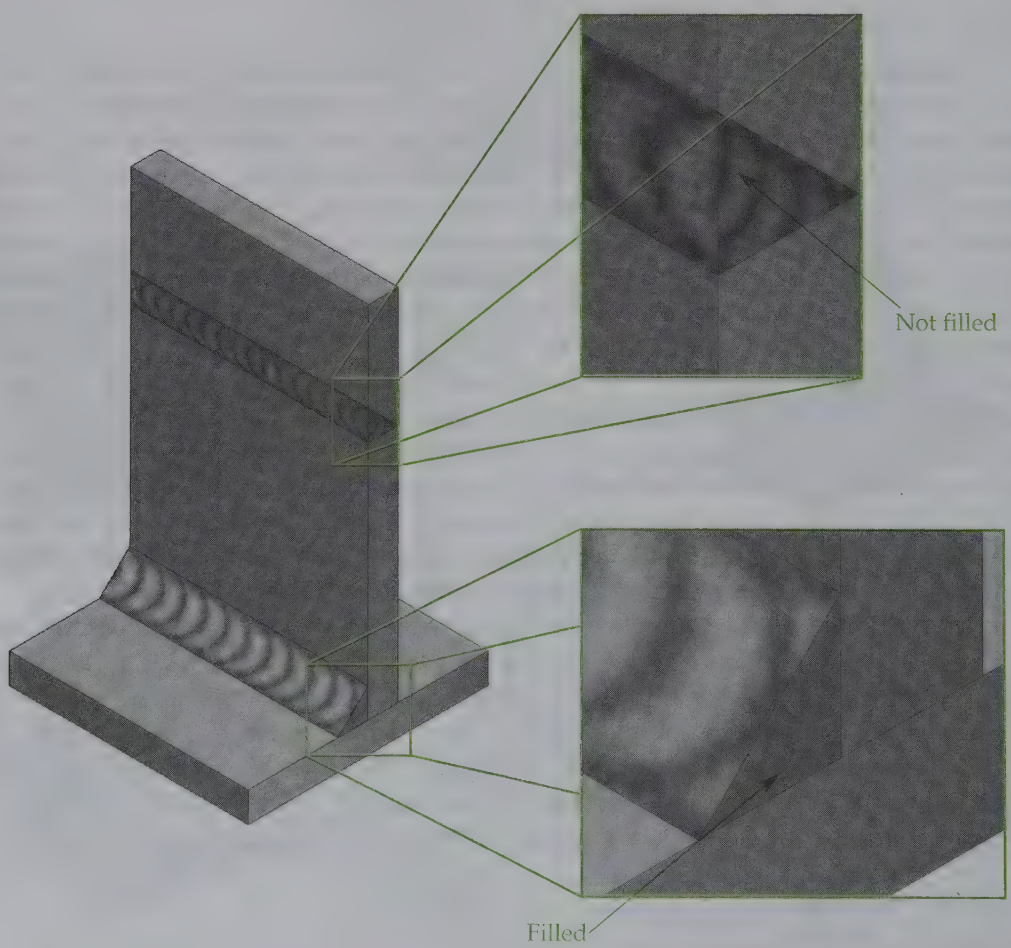
Figure 13-23.
The **Welding Symbol** dialog box.



Filling Ends

End fills identify the end of a weld bead. See Figure 13-24. An end fill added to a model is primarily used for representation purposes, but can be referenced in the weldment drawing to describe the end of a weld. Weld bead end fills are created using the **End Fill** tool, which you can access by picking the **End Fill** button on the **Weldment Features Panel** bar, or picking the **End Fill** button on the **Weldment Features Panel** toolbar.

Figure 13-24.
Compare the display of the weld bead end that is not filled to the one that is filled.



To use the **End Fill** tool, select the desired weld bead ends, as shown Figure 13-24. By default, when you create a fillet weld, an end fill is automatically applied to the weld bead. Consequently, if you pick the end of a fillet weld while using the **End Fill** tool, the end fill is removed. Groove welds, on the other hand, do not automatically generate an end fill. As a result, picking the end of a groove weld adds an end fill, while selecting the end again removes the end fill.

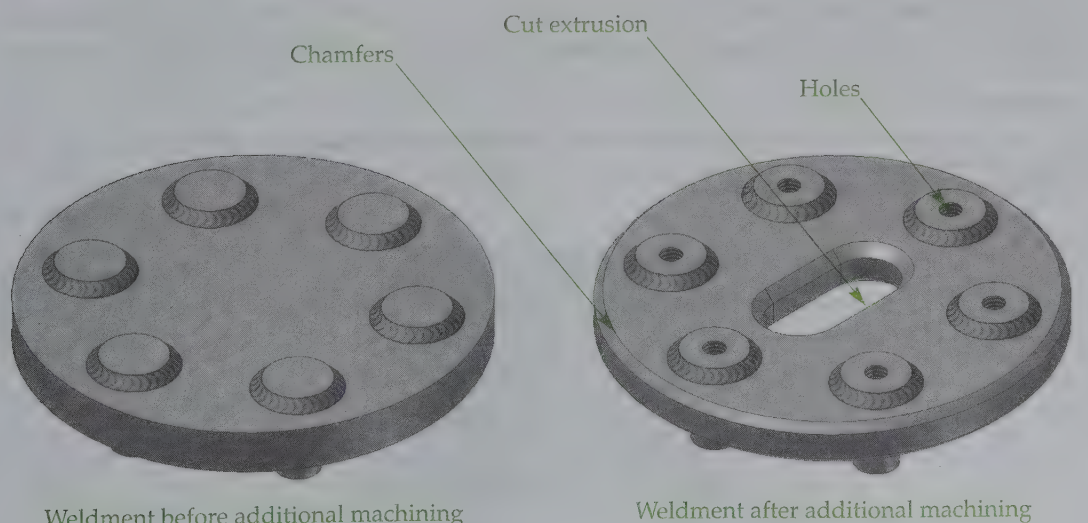
Machining Weldments

Weldment machining involves any task performed after you weld assembly components, such as drilling a hole through components, preparations, or welds. In many situations, additional machining may not be necessary. However, often a machined feature, such as a new joint, groove, hole, or other feature, is needed after welding. This may need to be completed before adding, preparing, and welding additional components. The type, location, and size of machined features you add to an existing weldment are specific to the welding application and design. Some simple weldment machining operations are shown in Figure 13-25.

Though you can machine individual weldment components in specific component files, Inventor provides tools for machining weldments while in the weldment file, much like a post-weld process performed on an actual weldment. Once you place or create weldment features, constrain the features to create an assembly, add preparations if needed, and weld components together, you are ready to machine the weldment. To enter the machining environment, double-click on **Machining** in the browser, or right-click on **Machining** in the browser and select the **Edit** pop-up menu option.

When you activate machining, you enter a simplified part feature environment that contains the **Extrude**, **Revolve**, **Hole**, **Sweep**, **Fillet**, **Chamfer**, **Move Face**, **Rectangular Pattern**, **Circular Pattern**, **Mirror Feature**, **Work Plane**, **Work Axis**, and **Work Point** tools. These weldment machining tools are used the same as in any other model environment, with the exception of the **Extrude**, **Revolve**, and **Sweep** tools, which only allow you to apply a cut operation, and the **Fillet** and **Chamfer** tools, which only allow you to remove material.

Figure 13-25.
Examples of various weldment machined features.

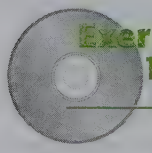


You can access various weldment-machining tools using many of the techniques described in the chapter where they apply. However, all weldment-machining tools are available from the **Weldment Features Panel** bar and the **Weldment Features Panel** toolbar.

Material is removed when you machine components after welding. As a result, Inventor provides the following machining options:

- Cut-extrude a sketch profile using the **Extrude** tool.
- Cut-revolve a sketch profile using the **Revolve** tool.
- Add a hole using the **Hole** tool.
- Cut a sketch profile along a sketch path using the **Sweep** tool.
- Place rounds using the **Fillet** tool.
- Place chamfers using the **Chamfer** tool.
- Move faces to add a root opening using the **Move Face** tool.
- Pattern features such as holes and slots using the **Rectangular Pattern** and **Circular Pattern** tools.

All machined features are stored in the **Machining** group in the browser. You can fully machine a weldment by entering the machining work environment once, or you may need to reenter **Machining** several times throughout weldment development. When you finish machining material, exit the machining work environment by double-clicking on the weldment name or icon, **Welds** or **Preparations** folder, or any of the weldment components in the browser; right-click and select the **Finish Edit** pop-up menu option; or pick the **Return** button on the **Inventor Standard** toolbar.

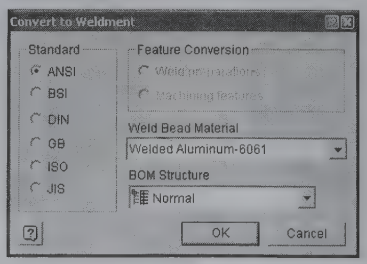


Complete the Exercise on the Student CD.

Converting an Assembly to a Weldment

As previously mentioned, you can convert an existing assembly into a weldment by opening the assembly you want to convert, and picking the **Weldment** option from the **Applications** pull-down menu. Again, once you transition from an assembly into a weldment, you will not be able to return to an assembly application. As a result, you may want to save a separate copy of the assembly before converting it to a weldment. When you select the **Weldment** option, you are prompted to continue with transition. If you select the **No** button, the conversion fails, but if you pick the **Yes** button, the conversion continues and the **Convert to Weldment** dialog box is displayed. See Figure 13-26.

Figure 13-26.
The **Convert to Weldment** dialog box.



The following information describes each of the **Convert to Weldment** dialog box options:

- **Standard.** This area contains radio buttons for all available standards. The standard radio button you select sets the weld symbols and specifications available when adding welds to the model. The standard radio button selected by default corresponds to the standard you specify in the template. However, you can use a different standard by selecting the desired radio button.
- **Feature Conversion.** This area controls the conversion of assembly features to weldment features. Pick the **Weld preparations** radio button to convert assembly features into weldment preparations, located in the **Preparations** folder in the browser. Pick the **Machining features** radio button to convert assembly features into weldment-machined features, located in the **Machining** folder in the browser.
- **Weld Bead Material.** Use this drop-down list to specify the material used to place welds. For example, if the weldment components are made from mild steel, you should select the **Welded Steel Mild** option.
- **BOM Structure.** This drop-down list is used to describe the status of the weldment in the bill of materials. The following information briefly describes each bill of materials structure option:
  - **Normal.** This option is selected by default and should be set for most standard components. A **Normal** status allows components to be displayed and recognized as assembly components, listed in the order specified in the assembly browser, and calculated according to the specified quantity.
  - **Inseparable.** Use this option to differentiate welded or otherwise permanently fastened subassembly components from other components. An assembly with an **Inseparable** status functions as a part in the **Parts Only** tab of the bill of materials.
  - **Purchased.** Pick this option to designate an assembly component as purchased. As the name implies, purchased items are bought from vendors and typically include standard parts and subassemblies, such as switches, fasteners, and springs. An assembly with a **Purchased** status functions as a part in the **Parts Only** tab of the bill of materials.
  - **Phantom.** This option is primarily used to reduce the number of components in the bill of materials, or to eliminate certain parts or subassemblies from the bill of materials. The components of an assembly with a **Phantom** status are still listed in the bill of materials unless the status of these child components is also redefined. A component with a **Phantom** status is not displayed in the bill of materials, but still exists in the model.
  - **Reference.** Assign this status to any components that are used only for reference purposes and have no other design or bill of materials value. **Reference** status components are not displayed in the bill of materials, and unlike the **Phantom** status, all child components associated with a **Reference** status are eliminated from the bill of materials.

Once you finish defining weldment conversion options, pick the **OK** button to generate the weldment subfile.



Complete the Exercise on the Student CD.

Editing Weldments

Weldments and weldment components are modified using many of the same techniques previously described in other modeling chapters. For example, once created, weldment components, constraints, preparations, welds, welding symbols, and machined features can all be redefined, if needed. In addition, the following information describes modification options specific to weldments and welds:

- **Weld visibility.** In some situations you may want to turn off the visibility of weld bead geometry. When weld beads are hidden, both the weld bead and the corresponding welding symbol are not displayed. To turn off weld geometry and symbol visibility, right-click on the **Welds** folder in the browser or on any weld in the graphics window, and deselect the **Visibility** pop-up menu option.
- **Weld symbol visibility.** Often when working with large or complex weldments, you may want to turn off the visibility of weld symbols, but not weld bead geometry. To turn off only weld symbol visibility, right-click on the **Welds** folder or a specific weld in the **Welds** folder in the browser or any weld in the graphics window and deselect the **Symbol Visibility** pop-up menu option.
- **Weld properties.** Weld properties control a variety of weld bead attributes, including material type, color, visibility, and selectability. To access weld properties, right-click on the **Welds** folder in the browser, or any weld in the graphics window, and select the **Properties** pop-up menu option to access the **Properties** dialog box. To change the weld bead material, pick the desired material from the **Material** drop-down list of the **Physical** tab. See Figure 13-27. Though you may already be familiar with the **Physical** tab of the **Properties** dialog box previously described, the **Weld Bead** tab, shown in Figure 13-28, is specific to working with weldments. The following options are available in the **Weld Bead** tab of the **Properties** dialog box:
  - **Visible.** Picking this check box displays weld bead geometry and weld symbols. Deselecting the **Visible** check box accomplishes the same task as right-clicking on the **Welds** folder in the browser, or any weld in the graphics window, and deselecting the **Visibility** pop-up menu option.
  - **Enabled.** This check box is selected by default, allowing you to select weld bead geometry. When the **Enabled** check box is deselected, you cannot select and edit welds.

Figure 13-27.
The **Physical** tab of the **Properties** dialog box.

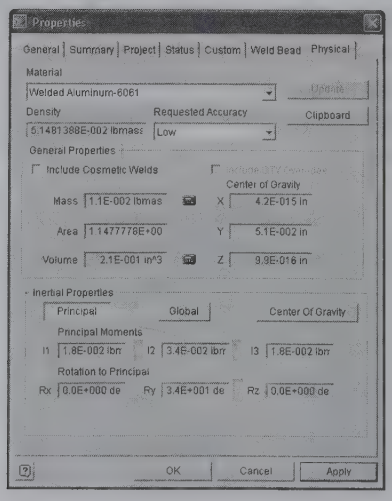
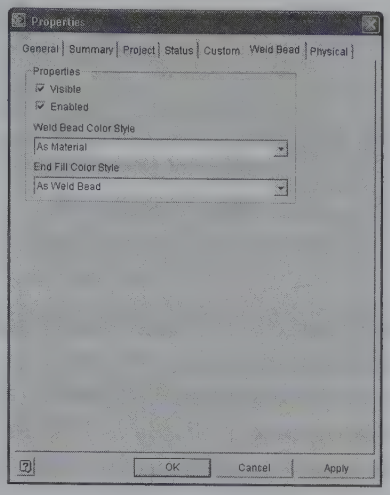


Figure 13-28.
The **Weld Bead** tab of the **Properties** dialog box.



- **Weld Bead Color Style.** This drop-down list controls the color and display characteristics of weld beads. In order to display the .bmp image of a weld bead and use the corresponding weld filler material, you must select the **As Material** color style option. However, if a weld bead image is not required, you can pick any desired color from the **Weld Bead Color Style** drop-down list. Another way to change the color of weld beads is to select the **Welds** folder in the browser, or any weld in the graphics window, and pick the desired color from the **Color Style** drop-down list of the **Inventor Standard** toolbar.
- **End Fill Color Style.** This drop-down list controls the color and display characteristics of weld bead end fills. You can pick any desired color from the **End Fill Color Style** drop-down list.

NOTE



Turning off weld bead or weld symbol visibility does not actually remove weld features or weld symbols from the model.

Creating Weld Bead Reports

Inventor includes the **Weld Bead Report** tool that can be used to export weld bead information as an .xls spreadsheet file. A weld bead report is a list of all the welds added to an assembly file and includes data on the Inventor tool used to create the weld bead and physical properties, such as the length of each weld, volume, and mass. See Figure 13-29. A weld bead report can be used to determine the volume and mass of weld beads and help calculate welding material usage, and weld times.

NOTE



Cosmetic weld beads do not store any physical properties that can be exported to a weld bead report.

Figure 13-29.
An example of a weld bead report.

	A	B	C	D	E	F	G	H	I	J	K
1	**Document**	**ID**	**Type**	**Length**	**UoM**	**Mass**	**UoM**	**Area**	**UoM**	**Volume**	**UoM**
2	C:\F13-29.iam										
3		Fillet Weld 2	Fillet	5	in	0.026	lbmass	0.018	in^2	0.092	in^3
4		Cosmetic Weld 2	Cosmetic	7	in	N/A		N/A		N/A	
5		Groove Weld 1	Groove	N/A		0.035	lbmass	N/A		0.125	in^3
6		Groove Weld 2	Groove	N/A		0.035	lbmass	N/A		0.125	in^3
7											

To create a weld bead report, access the **Weld Bead Report** tool by picking the **Weld Bead Report...** option from the **Tools** pull-down menu, or right-click on the **Welds** group in the browser and select the **Weld Bead Report...** option.

The **Weld Bead Report** dialog box is displayed when you access the **Weld Bead Report** tool. See Figure 13-30. If you want to include subassembly weld beads in the report, pick the **Include All Subassemblies** check box. Then, pick the **Next** button to access the **Report Location** dialog box shown in Figure 13-31. The **Report Location** dialog box is much like the **Save As** or **Export** dialog boxes explained throughout this book, but is used to export the weld bead information as a Microsoft Excel, .xls, spreadsheet file.

Figure 13-30.
The **Weld Bead Report** dialog box will appear during the creation of a weld bead report only when a subassembly exists in a weldment.

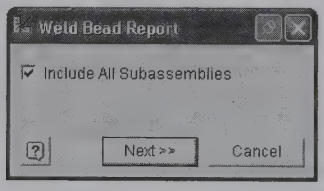
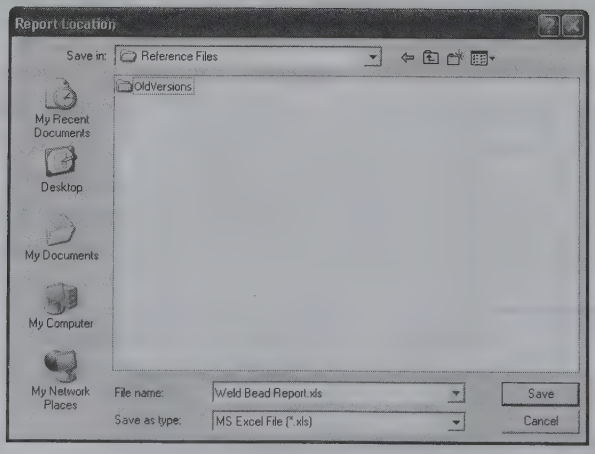
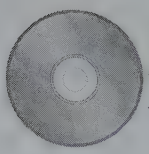


Figure 13-31.
The **Report Location** dialog box.





Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. Define *welding.*
2. Define *weldment.*
3. How is the weldment work environment different from and similar to the assembly work environment?
4. How are weldment templates listed by default?
5. Name the three additional groups, or nodes, which are similar to folders and contained in the weldment browser.
6. Explain what is involved in material preparation, and give an example.
7. Define *root opening.*
8. What does weld placement involve?
9. Explain the meaning of *arrow side.*
10. What does *other side* mean?
11. What do *leg, size,* and *depth* refer to?
12. Define *weld length.*
13. Define *weld pitch.*
14. Describe a *fillet weld* and identify the tool used to create it.
15. Define *weld bead leg.*
16. Describe the *weld bead throat.*
17. Discuss the relationship between the weldment model and the two-dimensional weldment drawing in regard to the welding symbol.
18. Describe the *reference line.*
19. Explain the function of a cosmetic weld.
20. What does a groove weld create?
21. Explain the difference between using the **Create Welding Symbol** check box and the **Welding Symbol** tool when adding welding symbols.
22. Provide an explanation of welding machining.
23. Explain why you may want to save a separate copy of an assembly before converting it to a weldment.
24. Discuss weld visibility and how you control the visibility of weld geometry and symbols.
25. Explain the function of the **Weld Bead Report** tool.

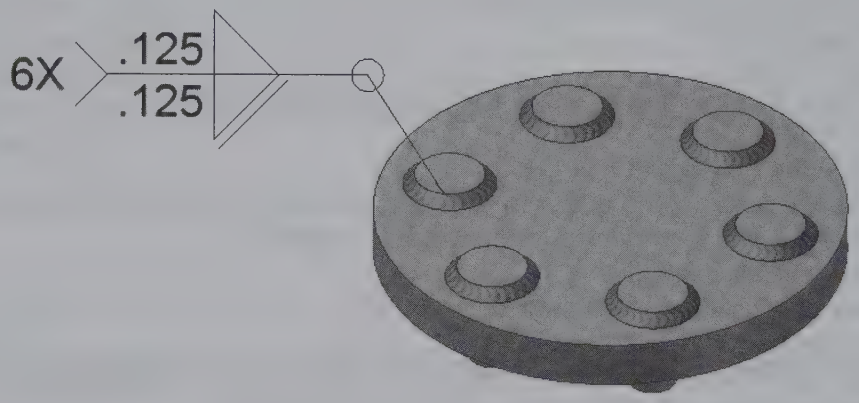
Problems

Instructions:

•Follow the specific instructions provided for each problem.

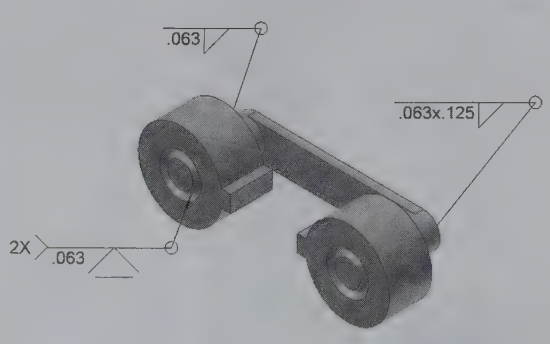
1. **Name:** Studded Hub
   Units: Inch
   Instructions:
   - Open EX12-17.iam, and save a copy as P13-1.iam.
   - Close EX12-17.iam, and open P13-1.iam.
   - Add the fillet welds shown. Notice that the welds are on both sides of the feature. Select the top and bottom face of the cylindrical plate as the first selection (two faces total), and then pick the large and small cylindrical stud faces as the second selection (12 faces total).

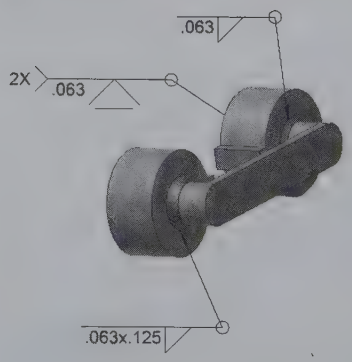
   Save As: P13-1.iam



2. **Name:** Alignment Bar
   Units: Inch
   Instructions:
   - Open EX12-11.iam, and save a copy as P13-2.iam.
   - Close EX12-11.iam, and open P13-2.iam.
   - Convert P13-2.iam to a weldment using the ANSI standard and Welded Stainless Steel, 440C weld bead material.
   - Place the four .0625" × .0625" chamfer preparations to create the V-grooves shown.
   - Add the cosmetic and fillet welds shown.

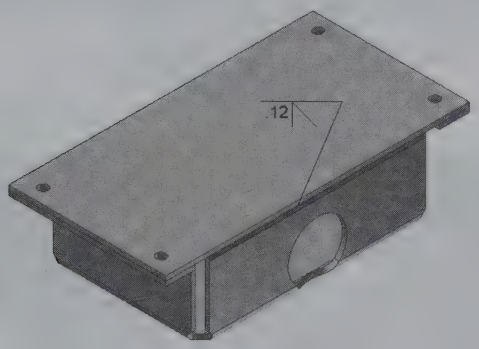
   Save As: P13-2.iam





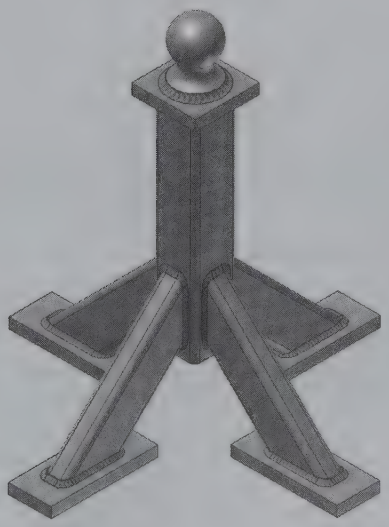
3. **Name:** Housing Assembly
Units: Inch
Instructions:
- Open P12-6.iam, and save a copy as P13-3.iam.
- Close P12-6.iam, and open P13-3.iam.
- Convert P13-3.iam to a weldment using the ANSI standard, and Welded Aluminum-6061 weld bead material.
- Place two .12" × .12" chamfer preparations to create the bevel-grooves shown. Place one chamfer on the edge of each side of the housing, not the housing cover.
- Add the groove welds shown to each side of the housing.

Save As: P13-3.iam

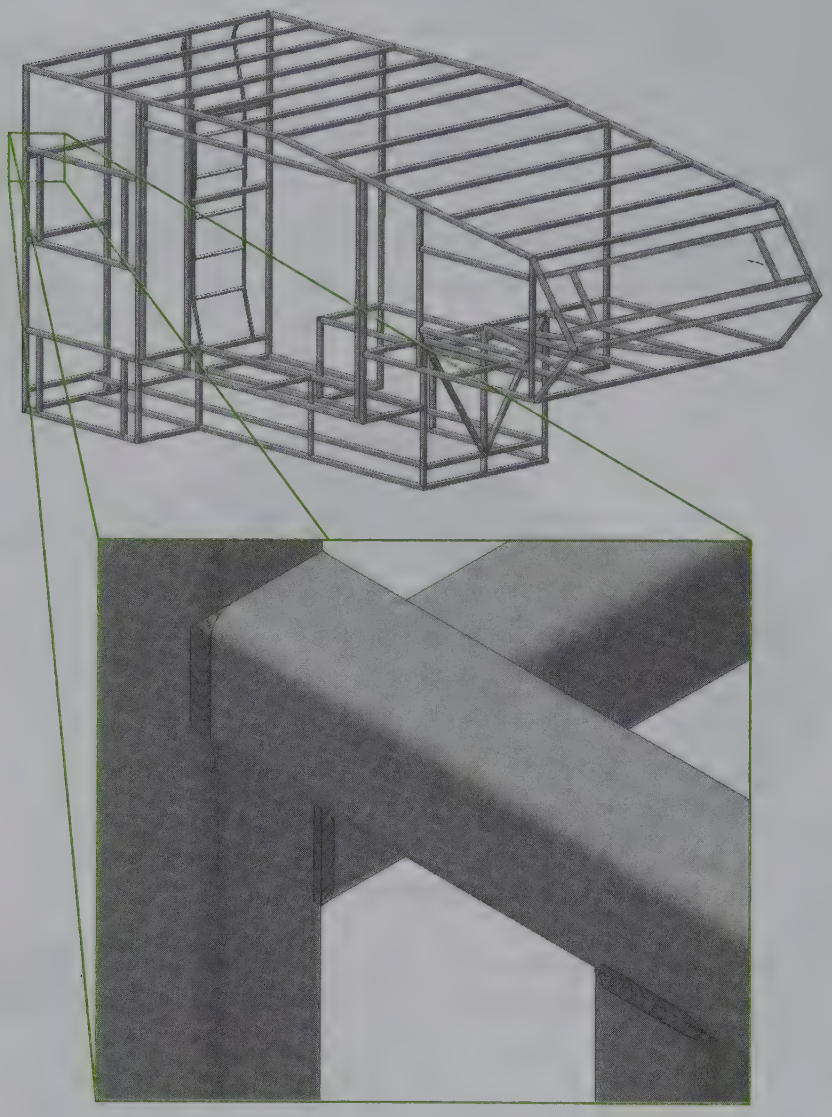


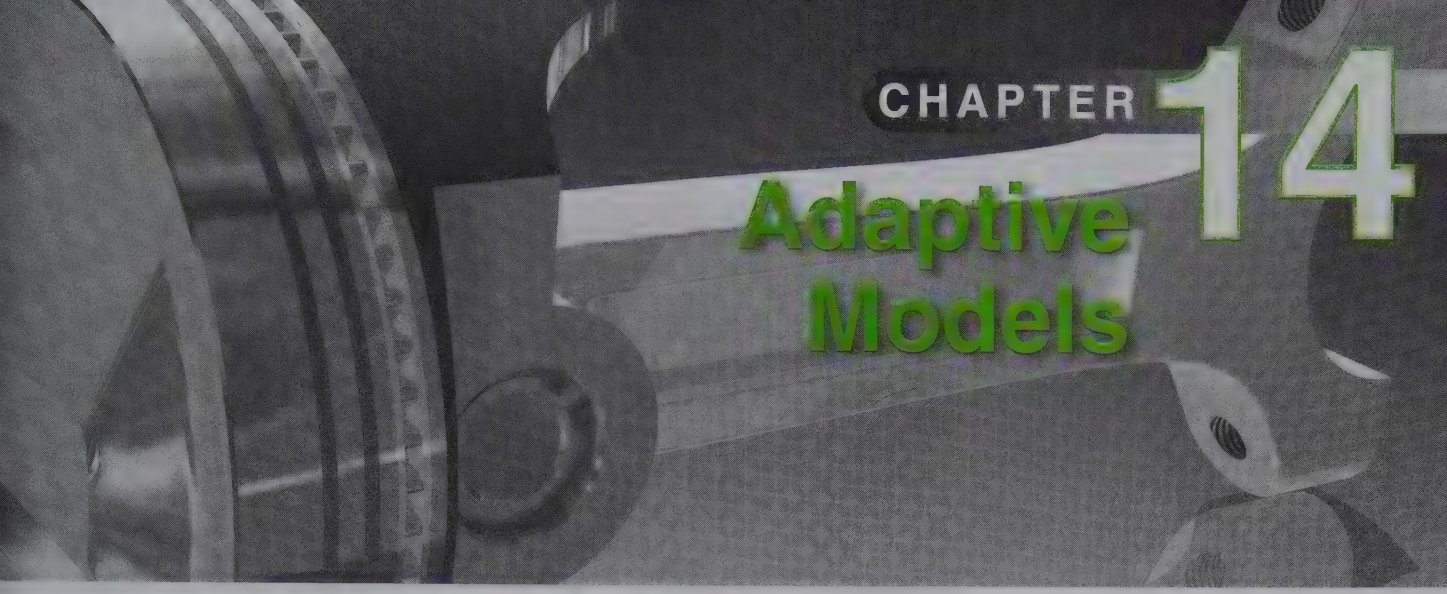
4. **Name:** Stand
Units: Inch or Metric
Template: WELDMENT-in or WELDMENT-mm
Instructions:
- Create the assembly shown using component dimensions of your choice.
- Add preparations, welds, and machined features as required, using your own specifications.

Save As: P13-4.iam



The detail shows some of the welds of this weldment





CHAPTER 14
Adaptive Models

Learning Objectives

After completing this chapter, you will be able to do the following:

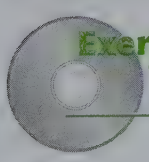
- Create adaptive sketches.
- Adapt sketches in-place.
- Develop adaptive features.
- Create adaptive assembly parts.
- Develop adaptive assembly subassemblies.

Often when developing a product, part and assembly geometry changes because of new design requirements. For example, you may need to increase the size of a hole in a certain part. Then, you must also increase the diameter of the shaft that fits inside the hole. Using this example, one option for defining a larger hole and shaft size is to individually edit the hole and shaft features and sketches in separate part files. However, Inventor contains adaptive capabilities that, in the previous example, automatically change the diameter of the shaft when the size of the hole is modified, or adjusts the diameter of the hole when the size of the shaft is redefined. ***Adaptive*** design elements change, or adapt, to modifications made to other associated assembly components. Adaptive items include part feature sketches, part feature parameters, and subassemblies. Adapting geometry is another way to define the location and size of model elements.

NOTE

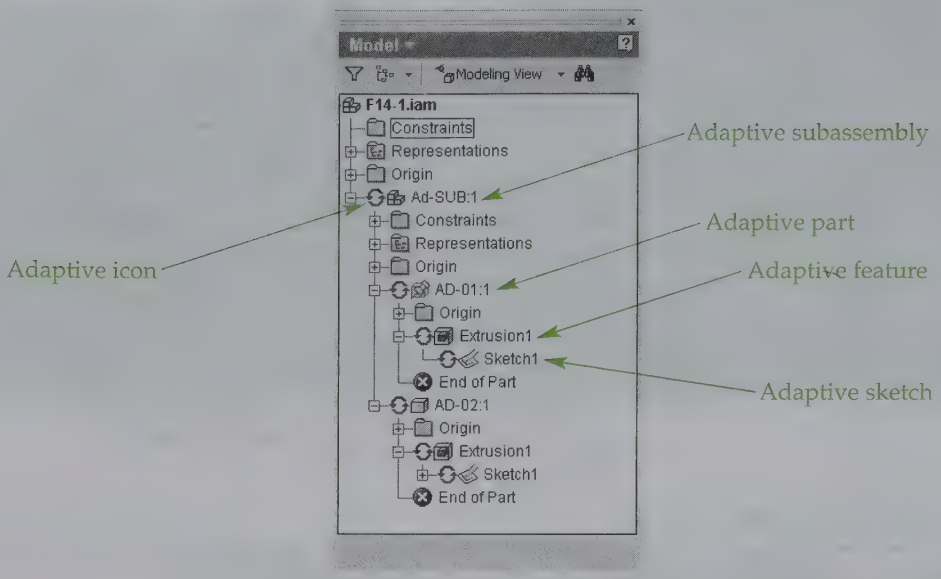


The browser displays the adaptive characteristics of sketches, features, parts, and subassemblies, using the **Adaptive** icon, shown in Figure 14-1.



Complete the Exercise on the Student CD.

Figure 14-1.
The browser display for an assembly with an adaptive subassembly, adaptive parts, and adaptive sketches.



Creating Adaptive Sketches

One option for creating adaptive assembly components is to use adaptive sketches that define sketched features. Sketches are constrained using geometric constraints and dimensions. Typically, you should fully constrain your sketches to ensure correct sketch dimensions and geometry. However, you may not want to completely constrain the sketch if you will not know the appropriate specifications of the sketched feature until the part is placed inside an assembly, or you want the sketched feature to adapt to other components. In these situations, an adaptive sketch is used. ***Adaptive sketches*** do not become fully constrained until they are placed inside an assembly and components are constrained together using assembly constraints.

The fully constrained sketch shown in Figure 14-2 cannot adapt to other geometry. When the sketch is extruded, only the extrusion can adapt, because all degrees of sketch movement have been removed. Adapting features is described later in this chapter. However, if one or more sketch dimensions or sketch constraints are not initially added or are removed, the sketched feature is free to adapt to other component geometry.

The key to developing an adaptive sketch and producing a sketched feature that can adapt to other component geometry is to not fully constrain the sketch.The sketch in Figure 14-3 does not contain any dimensions and only basic geometric constraints. When extruded and placed in an assembly, the sketch is free to constrain to other component geometry and create a part that is fully adaptive. However, most adaptive sketches have only one or two dimensions left off, allowing the part to adapt to specific component geometry. You must determine which sketch items you want to fully define and which you want to adapt, depending on the design intent and based on existing and future components. Typically, a sketch is used to create a sketch feature, which is often also adaptive. As a result, when a sketched feature becomes adaptive, the sketch automatically becomes adaptive. However, if you want to make a sketch adaptive, right-click on the sketch name in the browser or the sketch geometry in the graphics window, and select the **Adaptive** menu option.

Figure 14-2.
Removing a dimension or constraint in a sketch allows the part to adapt, change size, to a mated part in the assembly.

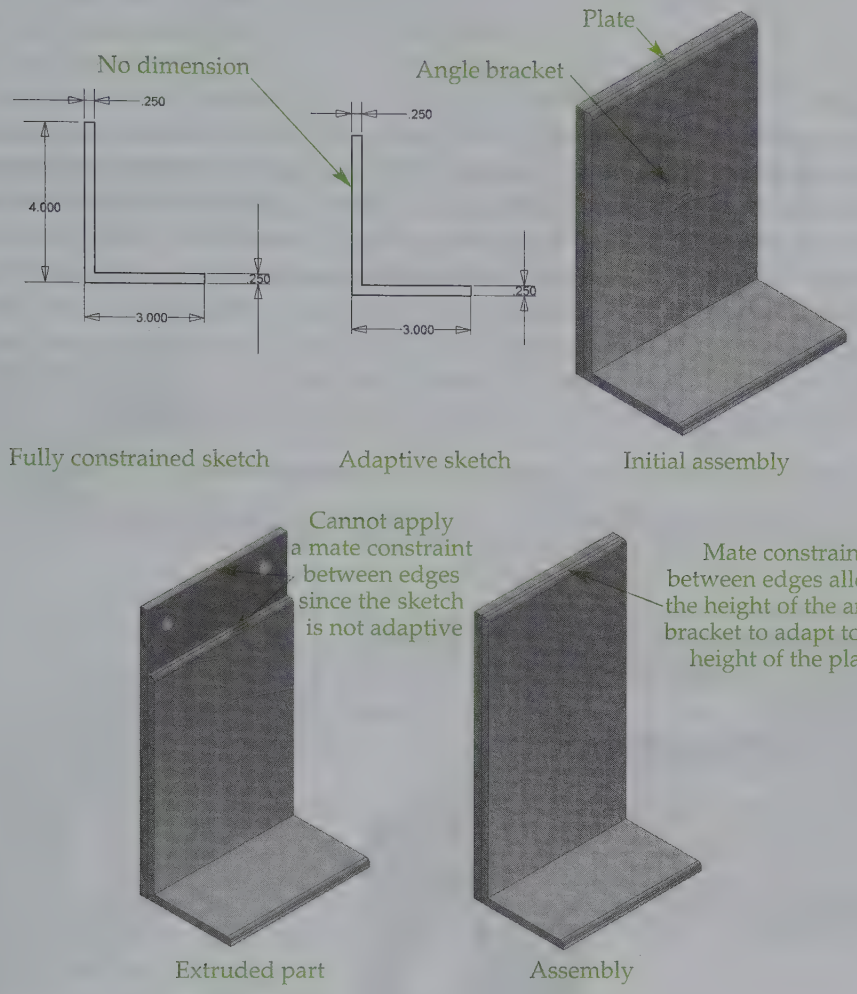
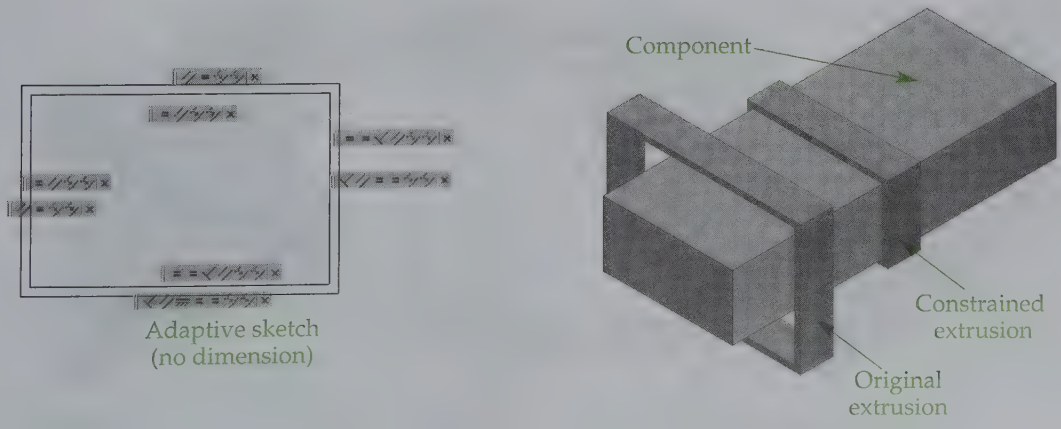
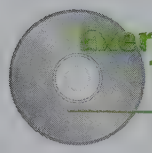


Figure 14-3.
Without any dimensions, this sketch is able to be adaptive and free to constrain to other components.



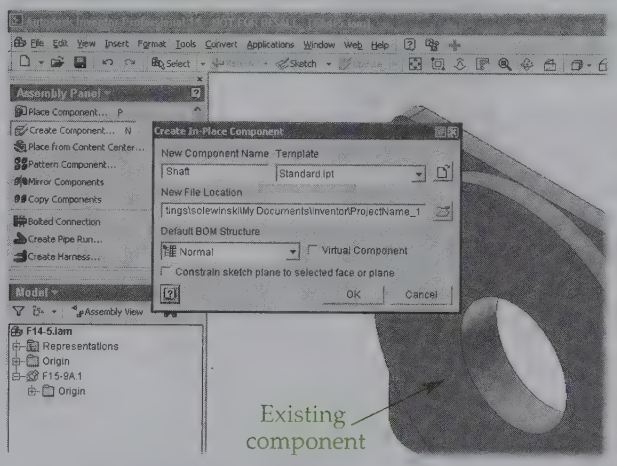


Complete the Exercise on the Student CD.

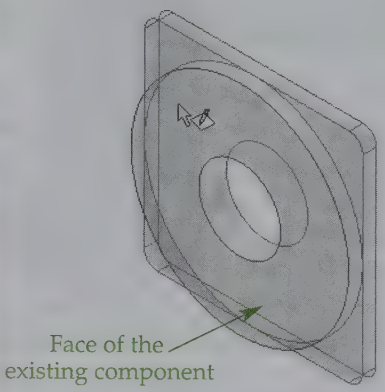
Adapting Sketches In-Place

In the assembly chapter of this book, you were introduced to adaptive sketches when creating new components in-place. You can use the **Create Component** tool to locate a new component sketch plane on an existing component face, and reference existing component feature geometry to define the new component. For example, if you want to add a shaft that is the same size as a part hole, place the new component sketch plane on the existing part face, and then project the hole's edge. This type of sketch is known as a ***reference sketch***. See **Figure 14-4**. When you use a reference sketch for developing a new component, the sketch and the new part automatically become adaptive, and an adaptive reference of the projected feature element is defined.

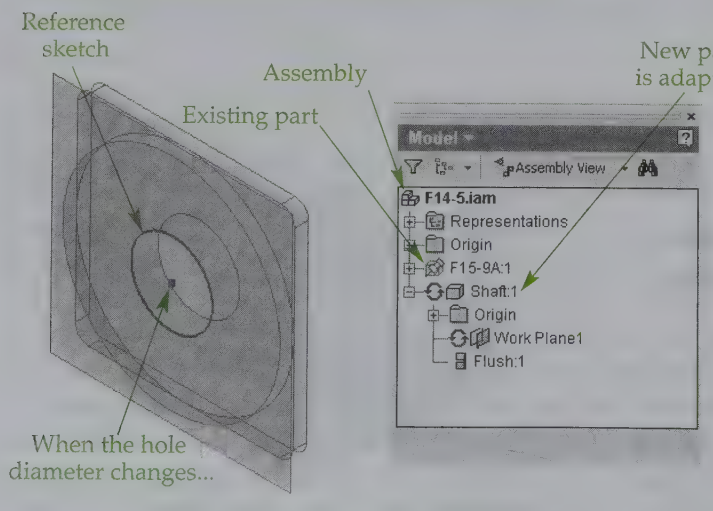
Figure 14-4.
A reference sketch can be used to create a component with a feature that changes size as the feature of another component changes size.



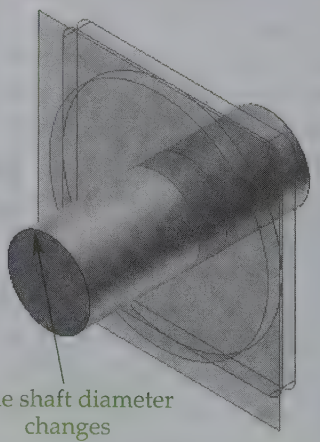
1. Create a new component in the assembly



2. Locate the sketch plane for the new component on a face of the existing component



3. Project geometry from the existing component to the new component



4. The extrusion created from this sketch is adaptive and references the existing part.

For many applications, a reference sketch is appropriate. However, you cannot dimension reference geometry, because it is essentially a piece of an existing component. You can remove the relationship between the existing parent feature and the new sketch by right-clicking on the reference in the browser and selecting the **Break Link** menu option. This action allows the sketch to become independent of the parent feature, even though the sketch is still referenced. Once the link between the sketch and the referenced component is broken, you can dimension and constrain the sketch, just like any other sketch.

When you finish a sketch created in-place using parent feature geometry, you will notice an assembly constraint is automatically placed between the existing component and the new sketch plane. See Figure 14-5. This constraint ensures the parametric, adaptive relationship between the existing component, the new component sketch plane, and the new component future face, edge, or point. The constraint is usually required to establish a link between two components, but can be removed if necessary.

PROFESSIONAL TIP

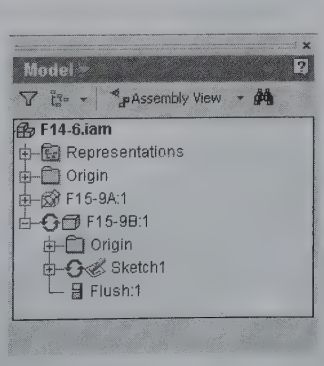


The **In-Place Features** area in the **Assembly** tab of the **Options** dialog box is used to specify preferences for creating components inside an assembly file. See Figure 14-6. Select the **Mate Plane** checkbox if you do not want features to be adaptive, but do want a mate constraint to occur between the feature and the plane. Pick the **Adapt Feature** checkbox to automatically adapt features to changing geometric planes. Choose the **Enable Associative Edge/Loop Geometry Projection During In-Place Modeling** checkbox to generate a sketch from selected geometry when creating a new feature.



Complete the Exercise on the Student CD.

Figure 14-5.
A constraint identifies the relationship between the sketch plane and the selected component face.



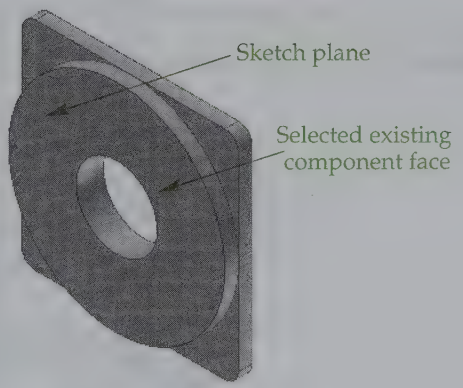
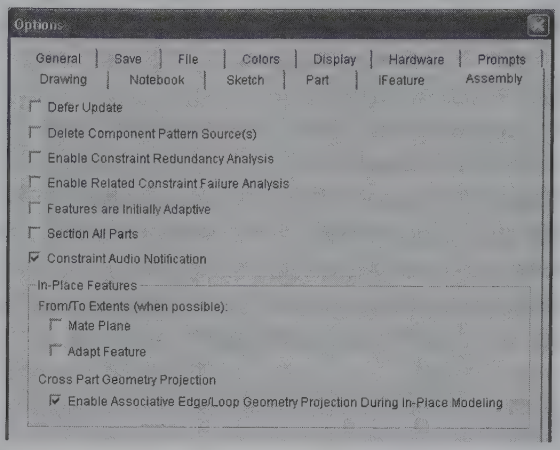


Figure 14-6.
The **In-Place** Features area contains settings for creating components inside an assembly file.

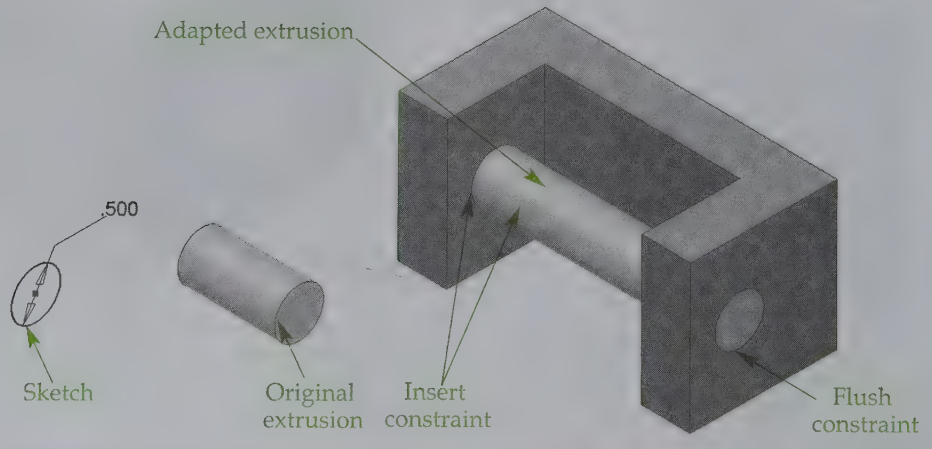


Developing Adaptive Features

Another way to adapt a part is by adapting the part feature. Often it is difficult to differentiate between an adaptive sketch and an adaptive feature because a sketch is required to create a sketched feature. Typically, an adaptive sketch is not fully defined using dimensions or geometric constraints. For example, you may choose to not add the width dimension to a sketched rectangle. This would allow the width of the rectangle, and the feature the rectangle creates, to adapt. As this example shows, when you adapt the sketch of a sketched feature, the feature is also adaptive. In contrast to an adaptive sketch, an adaptive feature may be fully defined. For example, if you extrude a 1″ × 3″ rectangle a distance of 5″, the extrusion is fully defined. However, if you adapt the extrusion using assembly constraints, the distance is free to change according to the size and location of assembly components.

You can adapt both a sketch and a feature together by not fully defining the sketch and by specifying the feature as adaptive. However, you do not have to create a sketch that is unconstrained and adaptive to adapt a feature. For example, an assembly might require a Ø.5″ rod, but the rod length is not defined until it is placed in the assembly, or the rod length changes to reflect a new design. Even though the adaptive rod is created by extruding a sketched circle, the sketch is fully defined using a Ø.5″ circle. See **Figure 14-7.**

Figure 14-7.
Working with an adaptive feature.



Most unconstrained geometry that is not fully defined can be made adaptive. The way these features adapt depends primarily on the design and assembly constraints. As previously mentioned, if you create a feature inside an assembly file that references existing component features, and the **Adapt Feature** checkbox in the **Options** dialog box is selected, the feature automatically becomes adaptive. In addition, if you set a sketch as adaptive, the corresponding sketched feature also automatically becomes adaptive.

Creating Adaptive Extrusions

To adapt an extrusion, right-click on the extrusion in the graphics window or the extrusion name in the browser and select the **Adaptive** pop-up menu option. When you use this option, the extrusion, the extrusion sketch, and the entire part become adaptive. Another method for adapting an extrusion is to right-click on the extrusion in the graphics window or the extrusion name in the browser, and select the **Properties** pop-up menu option. When you use this option, the **Feature Properties** dialog box is displayed. See Figure 14-8. This dialog box contains an **Adaptive** area, with the following checkboxes that allow you to specify the adaptive properties of the feature:

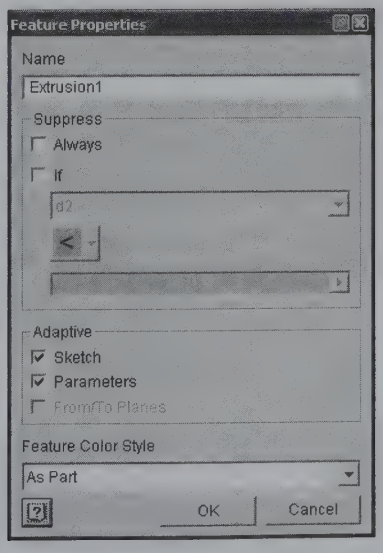
- **Sketch.** Pick this checkbox to specify the extrusion sketch as adaptive.
- **Parameters.** Select this checkbox to specify the extrusion parameters as adaptive. For example, if you define an extrusion distance, then select the **Parameters** checkbox, the distance becomes adaptive, allowing you to adapt the extrusion distance to other components.
- **From/To Planes.** Pick this button to adapt an extrusion that was created using the **From To** termination option.

NOTE



Only certain checkboxes are available inside the **Feature Properties** dialog box depending on the extrusion type.

Figure 14-8.
The **Feature Properties** dialog box for an extrusion.



Creating Adaptive Revolutions

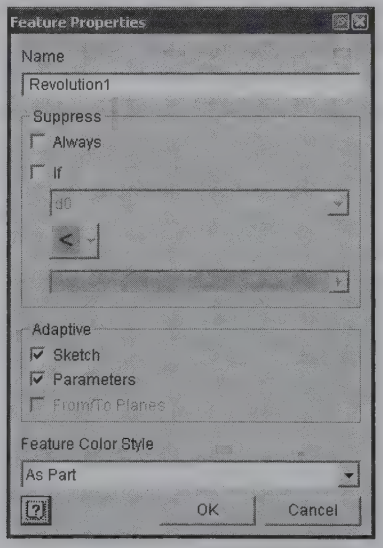
To adapt a revolution, right-click on the revolution in the graphics window or the revolution name in the browser and select the **Adaptive** pop-up menu option. When you use this option, the revolution, the revolution sketch, and the entire part become adaptive. Another technique for adapting a revolution is to right-click on the revolution in the graphics window or the revolution name in the browser and select the **Properties** pop-up menu option. When you use this option, the **Feature Properties** dialog box is displayed. See Figure 14-9. As with an extrusion, this dialog box contains an **Adaptive** area, with the following checkboxes, that allow you to specify the adaptive properties of the feature:

- **Sketch.** Pick this checkbox to specify the revolution sketch as adaptive.
- **Parameters.** Select this checkbox to specify the revolution parameters as adaptive. For example, if you define a revolution angle, select the **Parameters** checkbox and the angle becomes adaptive, allowing you to adapt the revolution angle to other components.

NOTE

The **From/To Plane** checkbox is not available for revolved feature properties, because **From To** is not a revolution termination option.

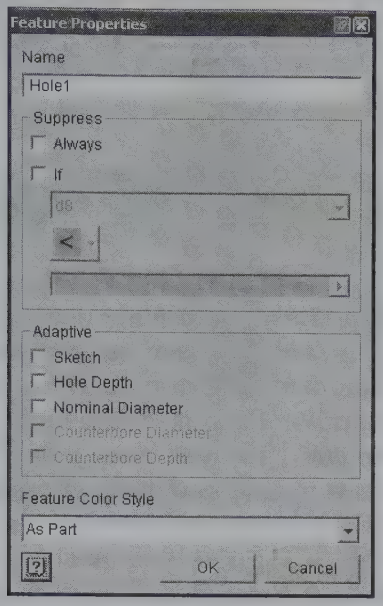
Figure 14-9.
The **Feature Properties** dialog box for a revolution.



Producing Adaptive Holes

To adapt a hole, right-click on the hole in the graphics window or the hole name in the browser and select the **Adaptive** pop-up menu option. When you use this option, the hole, the hole sketch, and all the hole parameters become adaptive. Another method for adapting a hole is to right-click on the hole in the graphics window or the hole name in the browser and select the **Properties** pop-up menu option. When you use this option, the **Feature Properties** dialog box is displayed. See Figure 14-10. This dialog box contains an **Adaptive** area, with the following checkboxes, that allow you to specify the adaptive properties of the feature:

Figure 14-10.
The **Feature Properties** dialog box for a hole.



- **Sketch.** Pick this checkbox to specify the hole sketch as adaptive.
- **Hole Depth.** Select this checkbox to adapt the depth of the hole.
- **Nominal Diameter.** Choose this checkbox if you want to adapt the nominal diameter of the hole.
- **Counterbore Diameter.** Pick this checkbox to specify the counterbore diameter as adaptive.
- **Counterbore Depth.** Select this checkbox to adapt the counterbore depth.

NOTE



Only certain checkboxes are available inside the **Feature Properties** dialog box depending on the type of hole you create.

Developing Adaptive Work Planes

To adapt a work plane, right-click on the work plane in the graphics window or the work plane name in the browser, and select the **Adaptive** menu option. Once you set features as adaptive, feature parameters are able to change in reference to other components and specified constraints. If a feature is not adaptive, you cannot specify assembly constraints in an effort to modify, or adapt, the feature. If a feature is not set as adaptive, you must edit the actual feature to modify its parameters, which does not necessarily ensure parametric relationships between certain components.

PROFESSIONAL TIP



The **Part Feature Adaptivity** area located in the **Assembly** tab of the **Options** dialog box is used to define the adaptivity of assembly components. Pick the **Features are initially adaptive** radio button if you want to automatically make new component features adaptive, or select the **Features are initially non-adaptive** radio button if you want to manually control the adaptivity of component features.



Complete the Exercise on the Student CD.

Creating Adaptive Assembly Parts

Parts are adapted in assemblies using assembly constraints. The assembly constraints you establish between adaptive parts and other assembly components depend on the application, design intent, and all other part and assembly model creation factors. Regardless of the application, in order for a part to adapt to other assembly components, the part must be adaptive. Even if you identify a part sketch or feature as adaptive in the part environment, you must also define the part as adaptive in the assembly environment. If a feature is not adaptive, it is known as a ***rigid body***. This means that you cannot specify assembly constraints in an effort to modify, or adapt, the feature. If a feature is not set as adaptive, you must edit the actual sketch or feature to modify its parameters, which does not necessarily ensure parametric relationships between assembly components.

To make a part adaptive in an assembly, right-click on the part in the graphics window or the part name in the browser and select the **Adaptive** pop-up menu option, or right-click on the part in the graphics window or the part name in the browser and select the **Properties** pop-up menu option. When you use this method, the **Properties** dialog box for the specific part is displayed. See Figure 14-11. Pick the **Occurrence** tab, and then select the **Adaptive** checkbox in the **Properties** area to make the part adaptive.

Assuming the part sketches and features are adaptive, once you specify part adaptivity, the part is free to constrain to other components. Figures 14-12 and 14-13 represent a few different examples of using adaptive parts. If you have difficulty constraining adaptive parts, particularly if you continually receive the "constraint is inconsistent with another" error message, one of the following problems may be the cause:

Figure 14-11.
The **Properties** dialog box for a part, **Occurrence** tab.

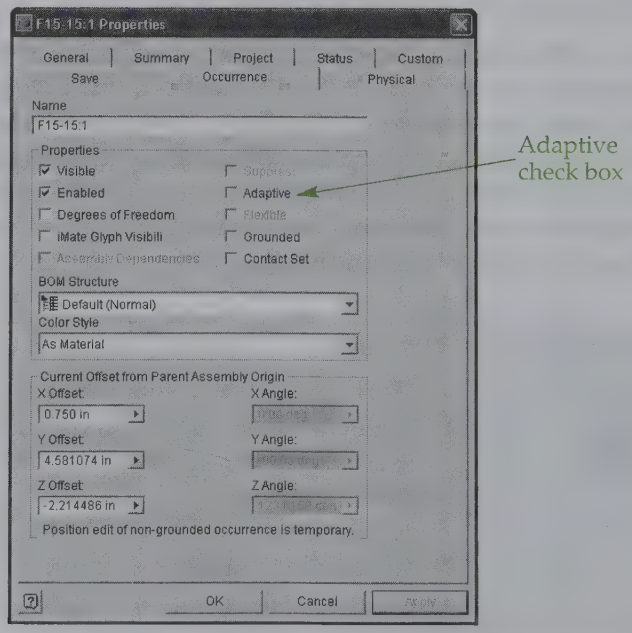
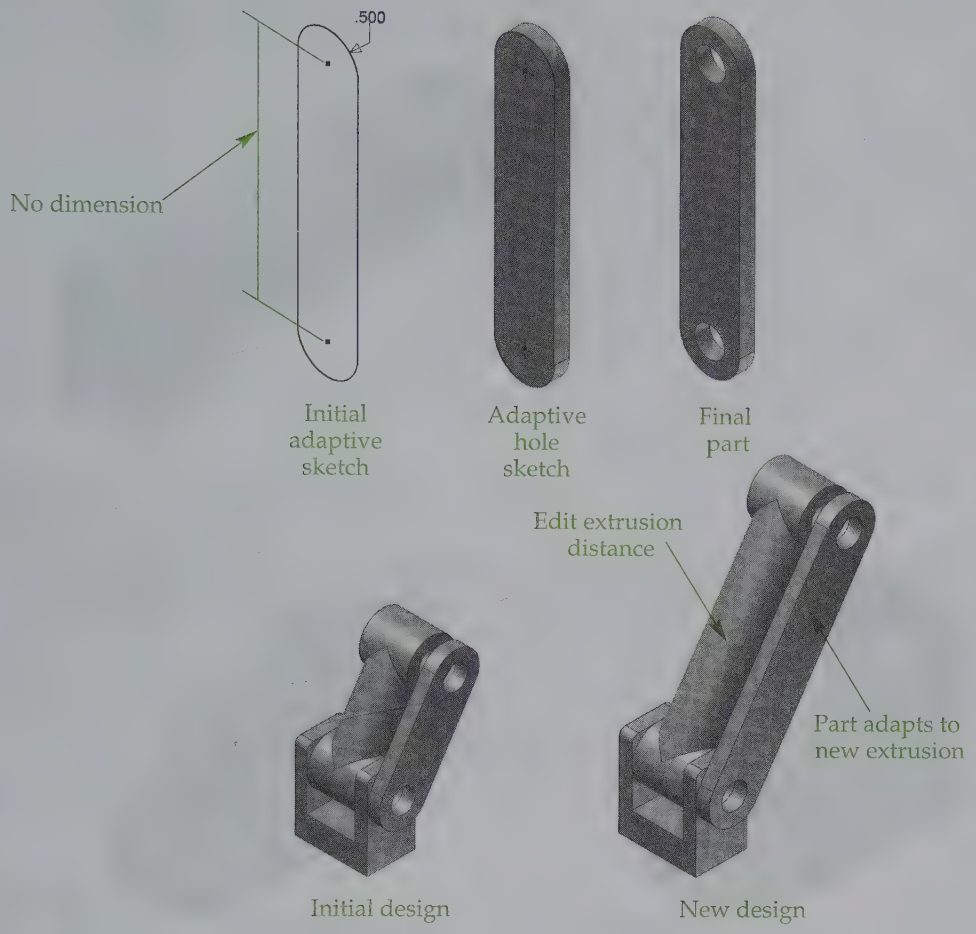


Figure 14-12.
Creating an adaptive part.



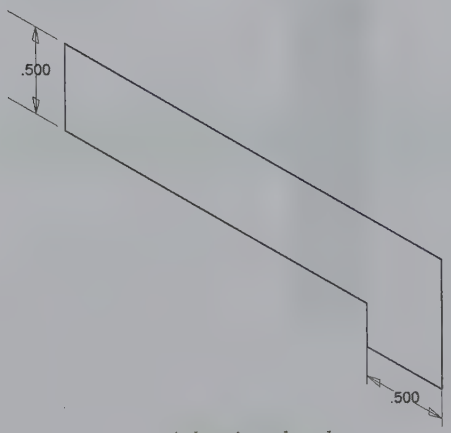
- You are trying to adapt a sketch that contains too many dimensions or geometric constraints. Solve the problem by removing the required dimension or geometric constraint.
- A sketch or feature of the part you are trying to constrain has not been made adaptive. Solve this problem by specifying the sketch or feature as adaptive.
- The wrong constraint is used. You may want to try an alternative constraint to solve the problem.
- The part has not been made adaptive. Adapt the part to solve the problem.

NOTE

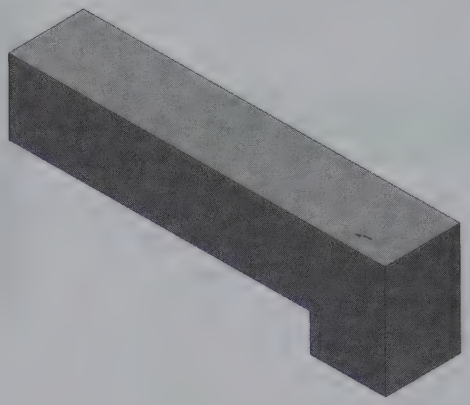


If an assembly does not change to reflect new constraints or relationships between adaptive parts, try an update. In some situations, more than one update may be necessary.

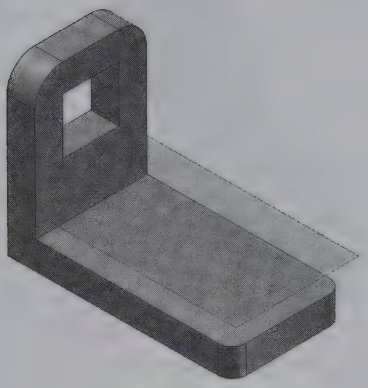
Figure 14-13.
Creating an adaptive part with an adaptive sketch and an adaptive feature. Adapting the part to another component using constraints.



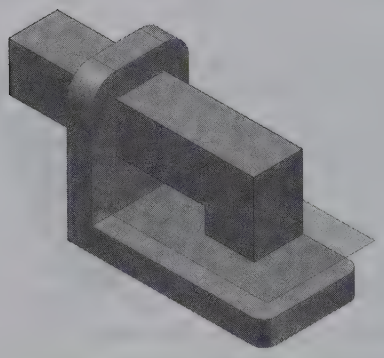
Adaptive sketch



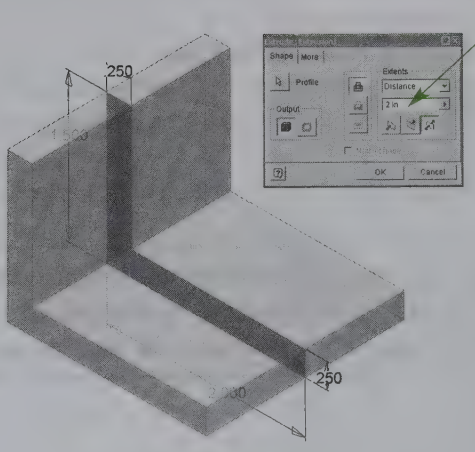
Adaptive feature



Nonadaptive part

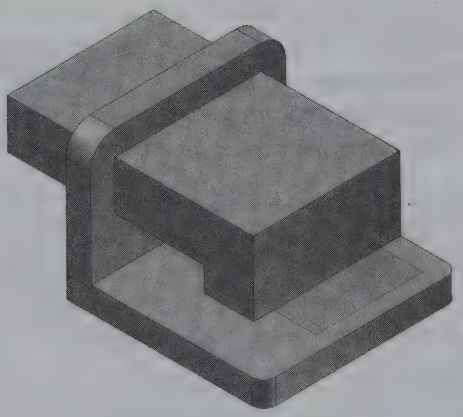


Assembly with mate constraints



Changing the size of the extrusion

Distance was 1 in.



Adaptive part adjusts to the change

PROFESSIONAL TIP



If an assembly contains several of the same parts, only one occurrence can be defined as adaptive and adjust to other component geometry. For example, if you insert two 120 mm × 35 mm × 50 mm blocks and adapt one of the blocks to change the length from 120 mm to 70 mm, both occurrences of the block change the length to 70 mm. In addition, because the parts are parametrically controlled, the separate part file will also reflect the change in length. To overcome this problem, save a copy of the original part, in this example, the 120 mm × 35 mm × 50 mm block. You may also want to save a separate copy of the part that is not adaptive.

PROFESSIONAL TIP



The **Adaptively Used in Assembly** checkbox, located in the **Modeling** tab of the part **Document Settings** dialog box should usually be picked, but you can deselect the checkbox if you do not want the geometry to be adaptive. For example, if you clear this checkbox while in a part file, then insert the part into an assembly, the part will not adapt, even if you have specified the part as adaptive.



Exercise 14-5 Complete the Exercise on the Student CD.



Exercise 14-6 Complete the Exercise on the Student CD.

Developing Adaptive Subassemblies

Subassemblies are adapted in assemblies using the same techniques as adapting parts. When you adapt a part in an assembly, a sketch and a feature must be adapted. In the same respect, when you adapt a subassembly in an assembly, the subassembly components must be adaptive, and the individual part sketches and features must be adaptive. In addition, subassemblies are also adapted using assembly constraints.

The constraints you establish between adaptive subassemblies and other assembly components depend on the application, design intent, and all other part, subassembly, and assembly model creation factors. Regardless of the application, in order for a subassembly to adapt to other assembly components, the subassembly must be adaptive. Even if you identify individual subassembly parts, part sketches, and part features as adaptive in the part environment and subassembly file, you must define

the subassembly as adaptive in the assembly environment. If a subassembly is not adaptive, the components are not free to move, and you cannot specify assembly constraints in an effort to modify, or adapt, the subassembly. If a subassembly is not set as adaptive, you must edit the actual sketches, features, parts, and constraints that define the subassembly, to modify its parameters, which does not necessarily ensure parametric relationships between certain components.

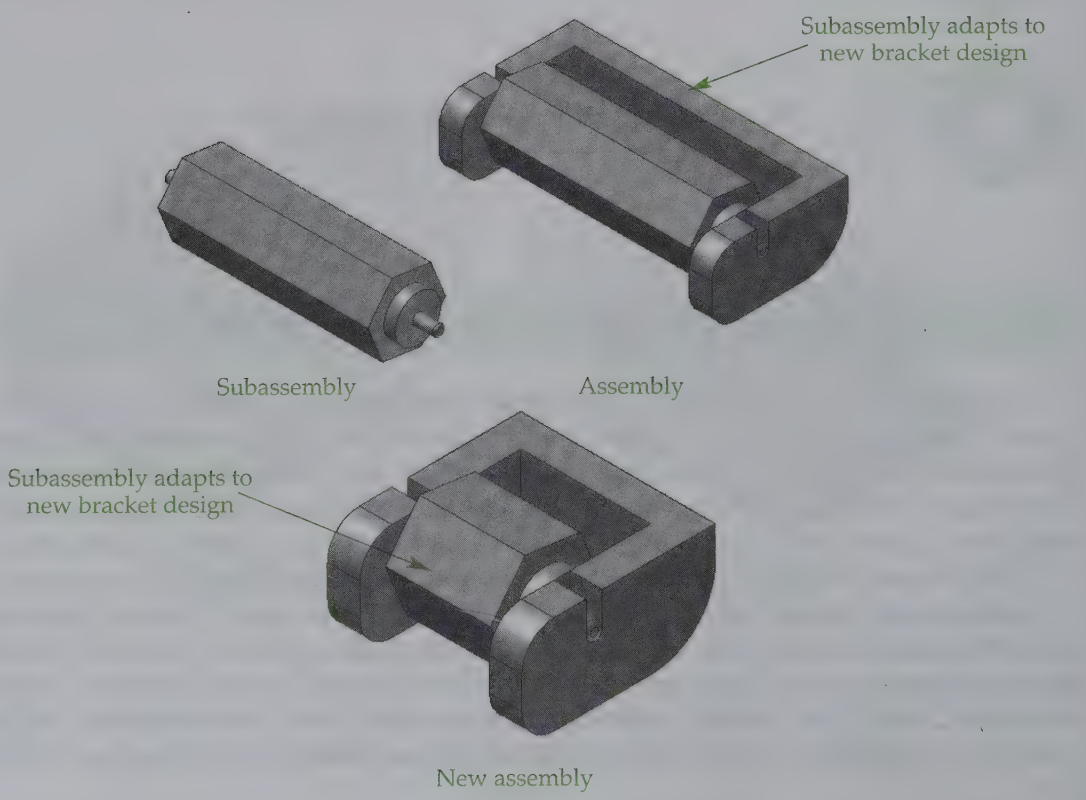
To make a subassembly adaptive in an assembly, right-click on the subassembly in the graphics window or the subassembly name in the browser and select the **Adaptive** pop-up menu option, or right-click on the subassembly in the graphics window or the subassembly name in the browser and select the **Properties** pop-up menu option. When you use this option, the **Properties** dialog box for the specific subassembly is displayed. Pick the **Occurrence** tab and then select the **Adaptive** checkbox in the **Properties** area to make the subassembly adaptive.

Assuming the individual part sketches and features are adaptive, and the subassembly parts are adaptive, once you specify subassembly adaptivity, the subassembly is free to constrain to other components. See Figure 14-14.

If you have difficulty constraining adaptive subassemblies, particularly if you continually receive the "constraint is inconsistent with another" error message, one of the following problems may be the cause:

- You are trying to adapt a sketch that contains too many dimensions or geometric constraints. Solve the problem by removing the required dimension or geometric constraint.
- A sketch or feature of one or more of the individual subassembly parts you are trying to constrain has not been made adaptive. Solve this problem by specifying the individual part sketches and/or features as adaptive.
- The wrong constraint is being used. You may want to try an alternative constraint to solve the problem.

Figure 14-14.
An example of adapting a subassembly.



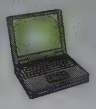
- A certain part or group of parts has not been made adaptive. Adapt the parts to solve the problem.
- The subassembly has not been made adaptive. To solve the problem adapt the subassembly.

NOTE



If an assembly does not change to reflect new constraints or relationships between adaptive subassemblies, try an update. In some situations, more than one update may be necessary.

PROFESSIONAL TIP



If an assembly contains several of the same subassemblies, only one of the occurrences adjusts to other component geometry. As a result, and because subassemblies are parametrically controlled, the separate assembly and individual part files also reflect changes. To overcome this problem, save a copy of the original subassembly and the individual parts, using different names. You may also want to save a separate unadaptive copy of the subassembly and subassembly parts.

NOTE



Setting a subassembly as flexible, removes the adaptivity of the subassembly.



Exercise 14-7 Complete the Exercise on the Student CD.



Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. Discuss how adaptive design elements function and give at least two examples of adaptive items.
2. Explain how adapting sketches, features, and subassemblies contribute to the parametric nature of Inventor, and give an example.
3. When do adaptive sketches become fully constrained?
4. What is the key to developing an adaptive sketch and producing a sketched feature that can adapt to other component geometry?
5. Define and discuss the function of a reference sketch and explain how a reference sketch and a new part automatically become adaptive.
6. How do you remove the relationship between the existing parent feature and the new sketch, so you can dimension and constrain the sketch, just like any other sketch?

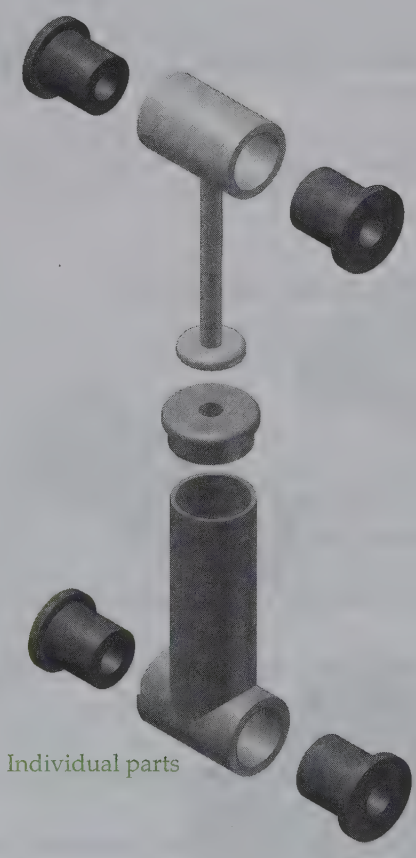
7. Identify two ways to make unconstrained geometry, which is not fully defined, adaptive. Include the function of the **Adapt Feature** checkbox in the **Options** dialog box in one of your options.
8. In order for a part to adapt to other assembly components, what must exist in the part and assembly environments for both to be adaptive?
9. What does *rigid body* mean?
10. Identify at least three reasons why you might have difficulty constraining adaptive subassemblies, particularly if you continually receive the "constraint is inconsistent with another" error message.

Problems

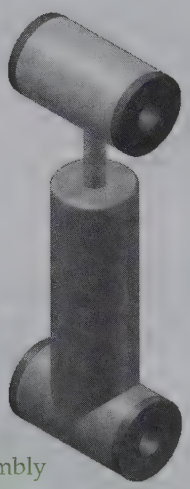
Instructions:

- Create parts, subassemblies, and assemblies similar to those shown.
- Using the information presented in this chapter, adapt the sketches, features, parts, and subassemblies as needed.
- Modify the assembly component sketches and feature parameters to observe the adaptivity.

1. **Name:** Shock Absorber Subassembly
   Units: Inch or Metric
   Template: ASSEMBLY-in.iam or ASSEMBLY-mm.iam
   Save As: P14-1.iam

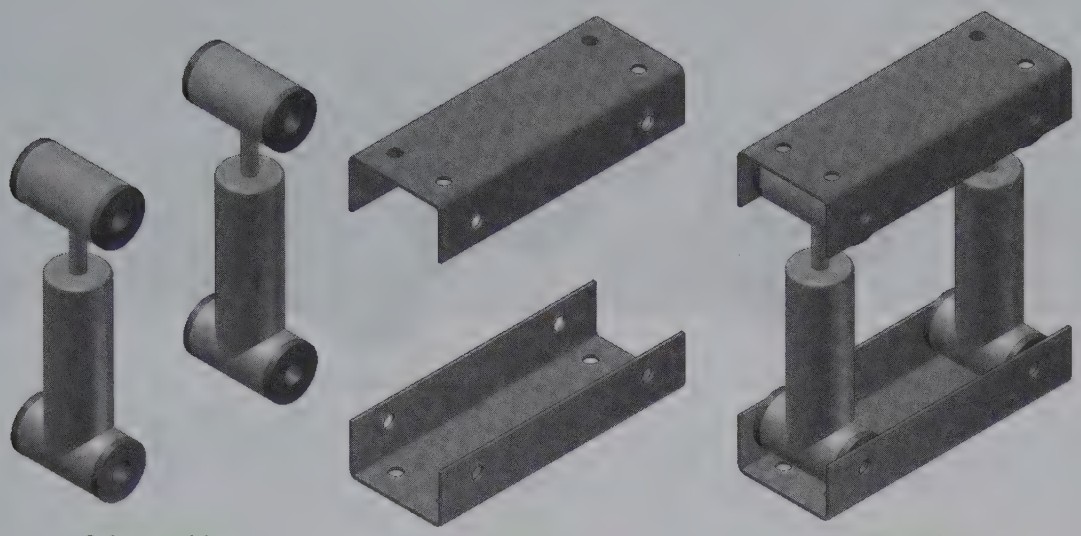


Individual parts



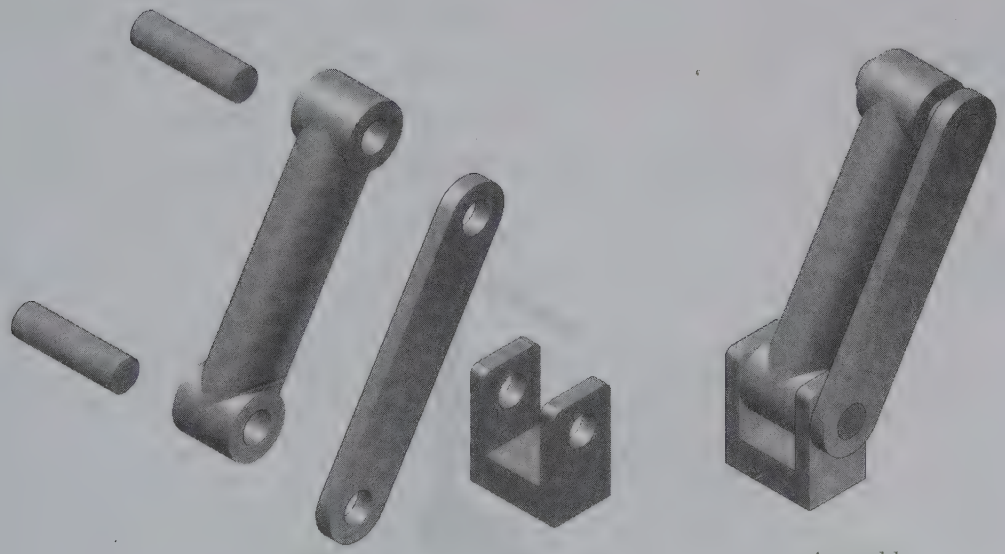
Assembly

2. **Name:** Shock Absorber Assembly
   Units: Inch or Metric
   Template: ASSEMBLY-in.iam or ASSEMBLY-mm.iam
   Save As: P14-2.iam



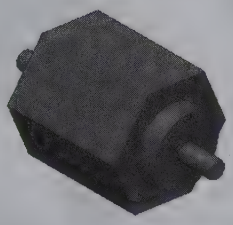
Subassemblies Parts Assembly

3. **Name:** Lever Assembly
   Units: Inch or Metric
   Template: ASSEMBLY-in.iam or ASSEMBLY-mm.iam
   Save As: P14-3.iam

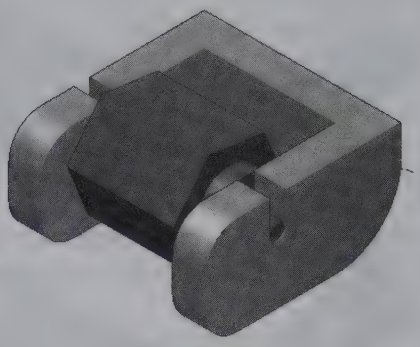


Parts Assembly

4. **Name:** Dispenser
Units: Inch or Metric
Template: ASSEMBLY-in.iam or ASSEMBLY-mm.iam
Save As: P14-4.iam



Subassembly



Assembly

CHAPTER 15

Presentations

Learning Objectives

After completing this chapter, you will be able to do the following:

- Create presentation views.
- Explode presentations.
- Tweak components.
- Edit tweaks and trails.
- Use the **Precise View Rotation** tool.
- Animate presentations.
- Place additional presentation views.

A ***presentation*** is a display of a complete assembly or weldment that can be exploded, animated, and used for two-dimensional presentation drawings. See Figure 15-1. A presentation is an ***exploded assembly***, which means that individual assembly components are moved away from other components in an unassembled or ungrouped fashion. Presentation views can be used to record design intent and help with assembly

Figure 15-1.
An example of a C-clamp presentation, created using a C-clamp assembly.



visualization. For example, a presentation clearly shows all assembly components that may be hidden in the assembly file due to constraints and assembly design parameters. A presentation can also be used to visually explain how assembly components fit together and interact with other components. Before you can make a two-dimensional presentation drawing you must create a presentation file.

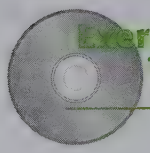
Developing Presentation Models

Inventor presentation files are separate files that carry the extension .ipn. Presentation template files are used to create presentations by inserting assemblies, weldments, and design views into a presentation file. You can create multiple individual presentation views in a single presentation file. For example, you may want to show a complete assembly in one presentation view, but only a couple components from the entire assembly in another presentation view. By default, when you open a new Inventor presentation template, the work environment should look similar to the display in Figure 15-2, and you are ready to insert an assembly file to create a presentation view using presentation tools. Presentation tools are used to develop presentations by inserting, exploding, rotating, and animating presentations. These tools are accessed from the **Presentation Panel** toolbar, or from the **Presentation** panel bar shown in Figure 15-2. Many presentation tools can also be accessed from the **View**, **Insert**, and **Tools** pull-down menus.

NOTE

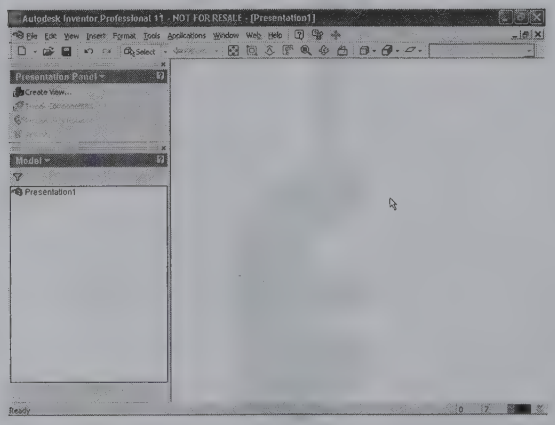


Presentation files are similar to assembly files, but do not contain assembly tools. You cannot modify component parameters and you do not have access to assembly options, such as work features, in the presentation environment.



Exercise 15-1 Complete the Exercise on the Student CD.

Figure 15-2.
The default Inventor presentation work environment.



Exercise 15-2 Complete the Exercise on the Student CD.

Exercise 15-3 Complete the Exercise on the Student CD.

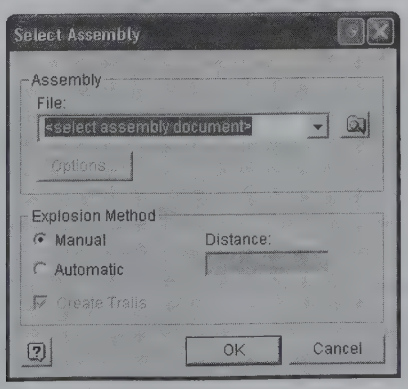
Creating Presentation Views

When you open a new presentation file (.ipn) you are ready to insert an assembly and generate a presentation view. The initial presentation view must be developed from an assembly file (.iam), because an assembly file identifies the model used for the presentation. When you create additional presentation views in the same presentation file, the new view references the initial assembly model. As a result, although you can place multiple presentation views in a single presentation file, the views must all come from the same assembly file. You cannot place different assembly files in a single presentation file.

To place the initial presentation view, access the **Create View** tool by picking the **Create View** button on the **Presentation Panel** bar; picking the **Create View** button on the **Presentation Panel** toolbar; selecting the **Create View...** option of the **Insert** pull-down menu; or right-clicking inside the graphics window or the browser, and selecting the **Create View...** pop-up menu option. When you access the **Create View** tool, the **Select Assembly** dialog box, shown in Figure 15-3, is displayed. The **Select Assembly** dialog box is similar to the **Open** dialog box, described in previous chapters, and allows you to locate the assembly you want to insert into the presentation file. In addition, the **Select Assembly** dialog box allows you to define the method of presentation explosion.

The **Assembly** area of the **Select Assembly** dialog box allows you to locate the assembly file, design view representation, positional representation, and level of detail representation you want to use to create the presentation view. To insert the initial presentation view, pick the **Browse** button next to the **File** drop-down list and locate the desired assembly file. Then, select a design view representation, positional representation, and level

Figure 15-3.
The **Select Assembly** dialog box.



of detail representation by picking the **Options...** button. When you select the **Options...** button the **File Open Options** dialog box is displayed. See Figure 15-4. The following options are found in the **File Open Options** dialog box:

- **Public.** Pick this radio button to select a design view representation stored in the assembly file in which you selected to insert into the presentation. Then, use the drop-down list to select the appropriate design view representation.
- **Private.** Choose this radio button to locate and use design view representations that are stored in a location other than the assembly file in which you selected to insert into the presentation. After you pick the **Private** radio button, select the **Browse** button to locate another existing .idv file. Then, use the drop-down list to select the appropriate design view representation.
- **Associative.** This checkbox is available once you select a design view representation from the drop-down list. Pick the **Associative** checkbox to link the assembly design view representation with the presentation, so that changes made to the design view are automatically applied to the presentation.
- **Positional Representation.** Use this drop-down list to select a predefined positional representation created in the assembly file, to use in the presentation.
- **Level of Detail Representation.** Use this drop-down list to select a predefined level of detail representation created in the assembly file, to use in the presentation.

Once you identify the assembly file you want to use to create the presentation view, the next step is to define the explosion characteristics using the **Explosion Method** area of the **Select Assembly** dialog box. If you pick the **Manual** radio button, you must manually explode the assembly components in the presentation view using the **Tweak Components** tool. Another option is to automatically explode the presentation when the presentation view is created by picking the **Automatic** radio button. When you select the **Automatic** radio button, the **Distance** edit box and **Create Trails** checkbox are available. Use the **Distance** edit box to enter the desired explosion distance for each component. An explosion distance is the tweak offset from the assembled presentation to the exploded presentation, and depends on how far you want exploded components to be spaced in reference to other exploded components. For example, the cylinder shown in Figure 15-5 has a tweak distance of 1". Select the **Create Trails** checkbox if you want trails to be displayed in the presentation. *Trails* identify the bond between individual components and the assembly by displaying the distance and location of the explosion. Once you select the desired assembly file and specify the explosion characteristics of the presentation, pick the **OK** button to create the presentation view.

Figure 15-4.
The **File Open Options** dialog box.

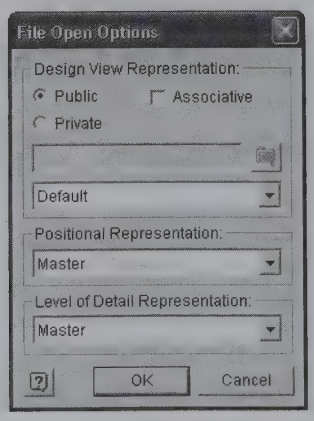
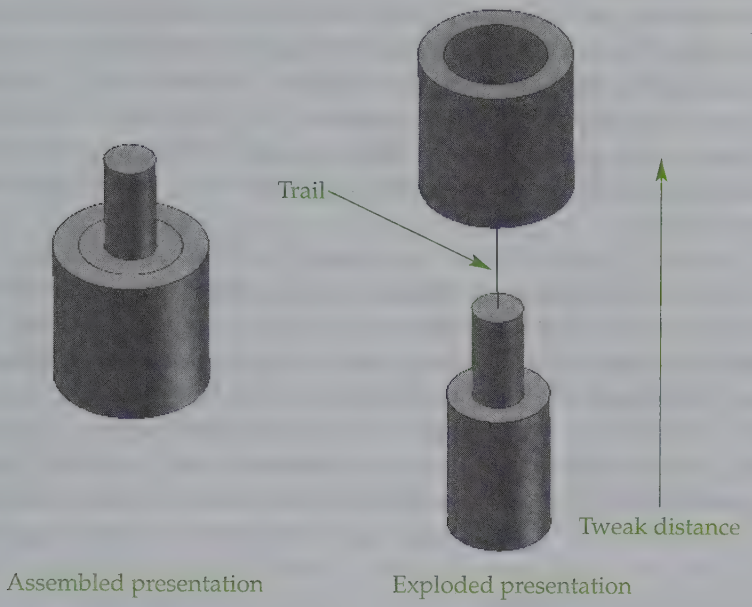


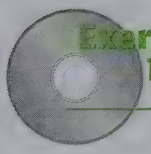
Figure 15-5.
This exploded presentation has a trail to indicate the distance and direction the components traveled.



PROFESSIONAL TIP



An automatically exploded presentation view display is based on assembly constraints. An automatic explosion may not display correctly, or may not occur at all, for large or complexly constrained assembly files. For these situations you will need to manually explode the presentation view, or modify the assembly constraints.



Exercise 15-4 Complete the Exercise on the Student CD.



Exercise 15-5 Complete the Exercise on the Student CD.

Exploding Presentations

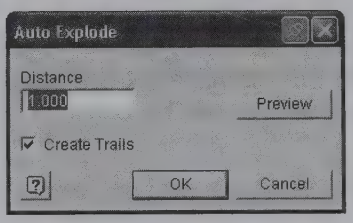
The main function of an assembly file is to develop an assembly of parts and subassemblies by constraining components to each other and defining a complete product. In contrast, one of the major purposes of a presentation is to display assemblies in a partially unconstrained, or exploded, fashion. ***Exploding*** an assembly in the presentation environment allows you to see all assembly components removed from,

and usually aligned with, each other. Just as presentation views have many purposes, based on your application, exploding an assembly in a presentation view, also known as an ***illustrated parts breakdown***, is useful for instruction manuals, identifying maintenance and assembly procedures, and creating assembly animations.

Inventor allows you to explode an assembly in the presentation view automatically or manually. An automatic explosion separates and aligns all the assembly components at one time, using a specified distance between the components. ***Tweaks***, which are modifications made to the originally constrained assembly, are automatically specified when you automatically explode. When you manually explode an assembly in the presentation file, you separate, or tweak, individual components from other components.

After creating a presentation view using the **Create View** tool, you can manually explode a presentation using the **Tweak Components** tool, or automatically explode a presentation. One option for automatically exploding the assembly is to select the **Automatic** radio button in the **Explosion Method** area of the **Select Assembly** dialog box. Refer to the previous section for information regarding this technique. Another way to automatically explode an assembly in a presentation view is to right-click on the assembly file name or assembly icon in the browser, and select the **Auto Explode** pop-up menu option. When you pick the **Auto Explode** menu option, the **Auto Explode** dialog box is displayed. See Figure 15-6. Exploding an assembly using the **Auto Explode** dialog box is the same as exploding an assembly when you insert the presentation view using the **Select Assembly** dialog box and the **Create View** tool. To use the **Auto Explode** dialog box to explode the view, enter the desired explosion distance for each of the components in the **Distance** edit box. Again, an explosion distance is the offset in which components are removed from other components. Next, select the **Create Trails** checkbox if you want to display presentation trails. As shown in previous figures, trails identify the relationship between component distance and location. Then pick the **Preview** button to display the intended explosion. If the preview looks acceptable, pick the **OK** button to apply the settings, and explode the assembly.

Figure 15-6.
The **Auto Explode** dialog box.





Exercise 15-6 Complete the Exercise on the Student CD.

Tweaking Components

It is often necessary to manually explode components. Manually exploding an assembly is done using the **Tweak Components** tool, which establishes presentation tweaks between selected components. To access the **Tweak Components** tool, pick the **Tweak Components** button on the **Presentation Panel** bar; pick the **Tweak Components** button on the **Presentation Panel** toolbar; press [T]; select the **Tweak Components...** option

of the **Insert** pull-down menu; right-click on an assembly or component in the browser and select the **Tweak Components...** menu option; or right-click on a component, or any location in the graphics window, and select the **Tweak Components...** menu option.

When you activate the **Tweak Components** tool, the **Tweak Component** dialog box, shown in Figure 15-7, is displayed. The **Tweak Component** dialog box allows you to identify the direction of separation, pick the components you want to move apart, and define trail and tweak specifications. The first step to tweaking a component is to identify the tweak direction by picking the **Direction** button if it is not currently selected. Then, as the direction icon shows, you can select any edge, face, or feature to define the desired direction of movement. See Figure 15-8. When you select an edge, face, or feature, one of the axis indicators is highlighted, indicating the axis is selected, and the component will move along the highlighted axis.

Once you identify the tweak direction, the **Components** button becomes active, allowing you to select the component or components you want to move. If you accessed the **Tweak Components** tool by right-clicking on a specific component or a component name, one component will already be selected. You still need to select any additional components you want to tweak at the same time. Conversely, you can deselect components by holding down the [Ctrl] key and picking the component you want to remove from the tweak. Figure 15-9 shows how a single or multiple components can be selected depending on the presentation application.

NOTE



In addition to selecting components, when the **Components** button is active, you can pick a different axis on the axis indicator to redefine the direction.

After you define the components you want to tweak, pick the **Trail Origin** button and select a point on the assembly to identify the origin, or beginning, of the trail. If you do not specify a trail origin, the trail will begin at the selected components' center of mass. If you do not want to show a trail, deselect the **Display Trail** checkbox.

The next step to tweaking components is to manage the following options in the **Transformations** area of the **Tweak Component** dialog box:

- **Triad Only.** Pick this checkbox if you want to rotate the axis indicator triad, shown in Figure 15-8, without actually rotating a component. To manipulate the axis indicator triad, ensure the **Triad Only** checkbox is selected, and then enter an angle in the **Distance/Angle** edit box. Finally, pick the **Apply** button to rotate the triad. You can also rotate the triad by holding down one of the axes and dragging/rotating the triad to the desired angle.

Figure 15-7.
The **Tweak Component** dialog box.

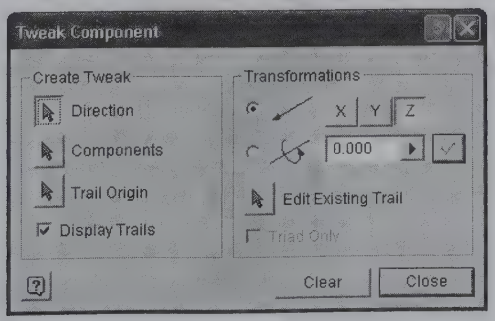
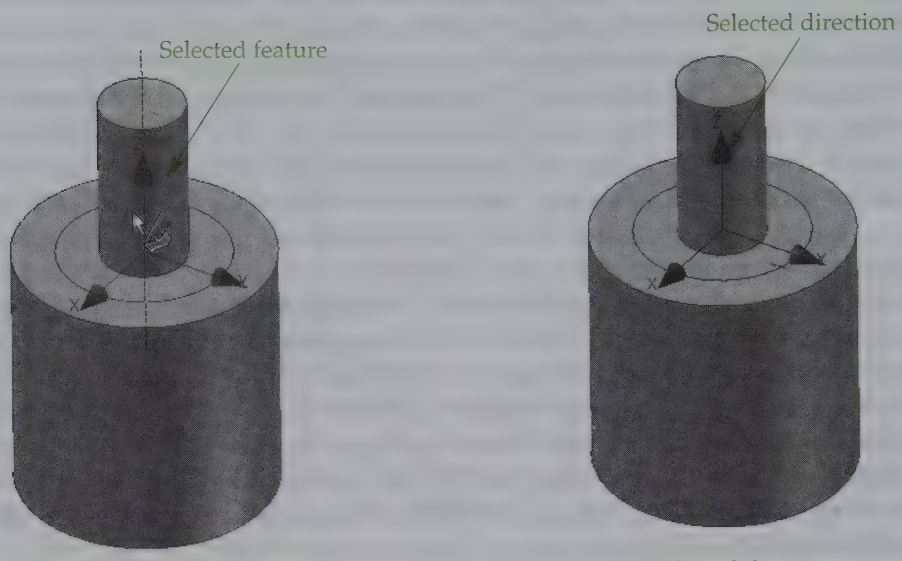


Figure 15-8.
The smaller cylinder was selected before selecting the Z-axis to specify the direction.



Selecting the tweak direction

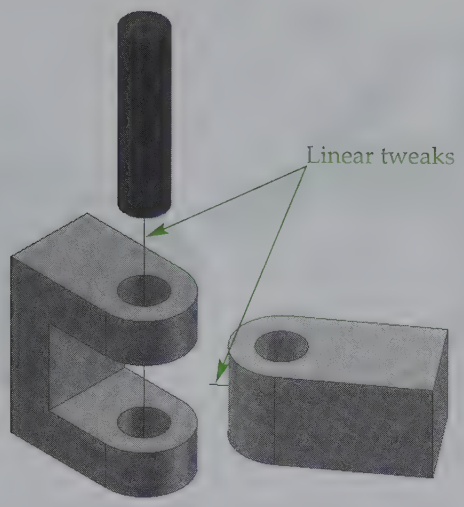
The selected direction

- **Linear Tweak.** Select this radio button if you want to move the component in a linear fashion along the specified axis. A linear tweak moves the component an identified distance. See **Figure 15-10.**
- **Rotational Tweak.** Pick this radio button to define a rotational tweak, which rotates the component a specified angle. A rotational tweak is often used with a linear tweak anytime you want a component to revolve around the selected axis and travel, such as when a nut threads onto a bolt. See **Figure 15-11.**

Figure 15-9.
The screw and pin were tweaked in the vertical direction, and then the pin was tweaked in the horizontal direction.



Figure 15-10.
Examples of linear tweaks.



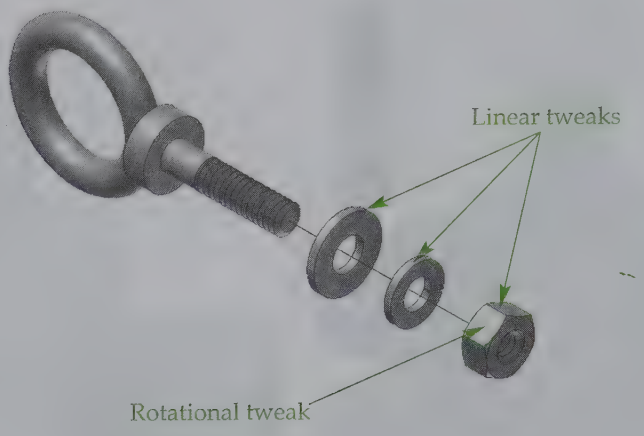
- **Direction.** Using these buttons is another way to adjust the specified tweak direction. The axis button that is currently selected is activated, but you can choose the **X**, **Y**, or **Z** buttons, depending on the tweak axis you want to define.
- **Distance/Angle.** If you are creating a linear tweak, use this edit box to enter the distance you want the component to be offset from the assembly and other components. If you are creating a rotational tweak, use this edit box to enter the angle of rotation. For example, if you want a hub to fully rotate, enter a value of 360°.
- **Apply.** Pick this button to create the specified tweak. When you pick the **Apply** button, the **Tweak Component** dialog box remains open, allowing you to continue applying tweaks.
- **Clear.** Select this button to reset the **Tweak Component** dialog box, so you can apply an additional tweak.
- **Close.** Once you specify the desired number of tweaks, pick this button to close the **Tweak Component** dialog box.
- **Edit Existing Trail.** Select this button to modify an existing tweak trail. Once you pick the **Edit Existing Trail** button, select the trail you want to modify, and then edit the distance or angle by dragging the component trail, or enter a different value in the **Distance/Angle** edit box.

NOTE



The **Assembly View** browser filter option is set as default and displays the assembly hierarchy, which shows the presentation view, followed by the assembly file, and each of the assembly components. In addition, tweaks are identified under the specific components. Select the **Tweak View** browser filter option if you want to display a tweak hierarchy to clearly identify presentation tweaks, which specify the explosion characteristics of the presentation. The tweak specifications, used to explode the assembly in each presentation view, are displayed first, and then the assembly and assembly components are shown.

Figure 15-11.
A rotational tweak can be used to show a nut rotating as it is threaded on a bolt.



PROFESSIONAL TIP

Although entering a specific distance ensures a linear tweak is created using a specific offset, you can hold down a component and linearly drag it to the desired location. You will notice the **Distance** edit box will identify the distance you drag the component.

Exercise 15-7 Complete the Exercise on the Student CD.

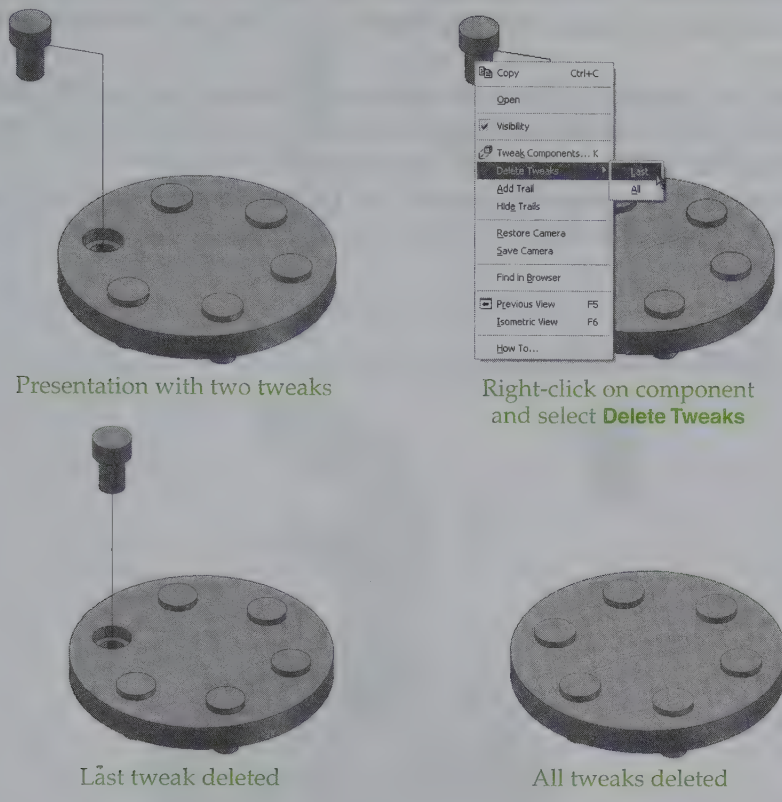
Editing Tweaks and Trails

You must often edit presentation parameters once they are defined. Typically, the only presentation factors that may require modification are tweaks and trails. The location of the available tweak and trail editing options depends on the specified browser filter setting. In addition, if you define a **Sequence View** filter, you can modify tweak sequence parameters that are not available when the **Assembly View** filter is set. However, as previously mentioned, sequence parameters are typically used for animation purposes. This section identifies specific tweak and trail editing options available in the browser, but also other, more general tweak and trail modification tools.

When you right-click on a component in the graphics window, or in the browser, that has been tweaked, the following edit options are available:

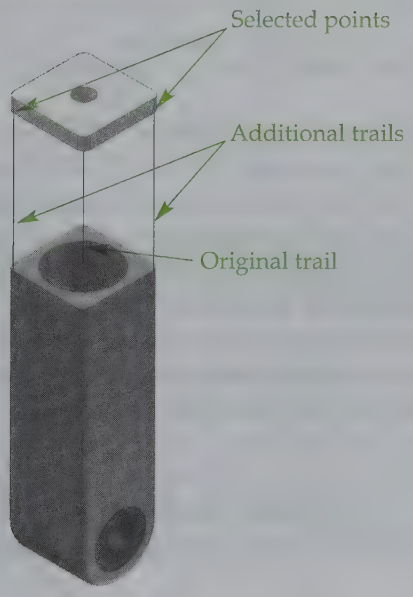
- **Delete Tweaks.** This option displays a cascading submenu that allows you to delete tweaks. See Figure 15-12. If a component has more than one tweak, and you want to delete the most recently created tweak, select the **Last** cascading submenu option. If you want to remove all tweaks associated with a component, pick the **All** cascading submenu option.
- **Add Trail.** Use this option to display an additional tweak trail. Once you select the **Add Trail** option, pick one or more points on the component. Each point you

Figure 15-12.
To remove the last or all tweaks, right-click on the component and select **Delete Tweaks**.



select will create a new trail. The only points you can select are on the specified component, because the trail is automatically referenced to other assembly components. To redefine the point or points you select, right-click and choose the **Reselect** menu option, and pick the new point. When finished selecting points for new trails, right-click and select the **Done** menu option to generate the new trail or trails. See Figure 15-13.

Figure 15-13.
Additional trails can be added to a tweak for clarity.

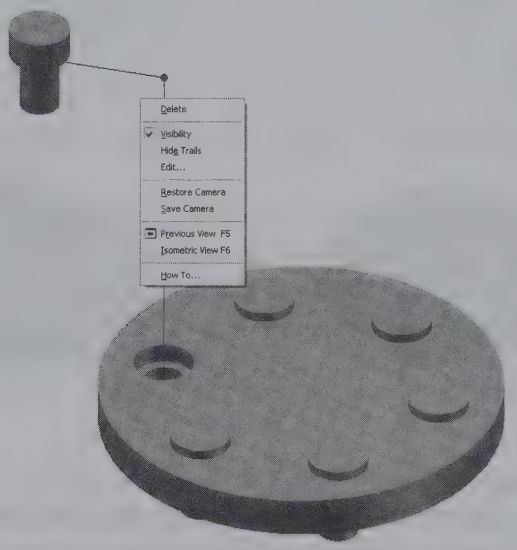


- **Hide Trails.** Select this option to "turn off," or hide, tweak trails so they are not visible.

When you right-click on a tweak trail in the graphics window, the following edit options are available, including the **Hide Trails** option previously described, as shown in Figure 15-14:

- **Delete.** This option functions similarly to the **All** selection of the **Delete Tweaks** cascading submenu, and removes the entire tweak, not just the trail.
- **Visibility.** This option "turns off" the selected tweak trail.
- **Edit...** Select this option to reopen the **Tweak Component** dialog box for the specified tweak, allowing you to redefine tweak parameters.

Figure 15-14.
Right-click on a tweak trail to display this pop-up menu.



NOTE

You can also open the **Tweak Component** dialog box for a specified component tweak by double-clicking on a component or component trail. The **Delete** and **Visibility** options are also available when you right-click on a tweak in the browser.

When you access the **Tweak Component** dialog box, you can modify tweak parameters. However, if all you want to do is extend the tweak distance or modify the rotational angle, you can use one of the following options:

- **Drag.** To change the explosion offset, pick the trail of the tweak you want to modify. Then drag the green dot to the desired location. See Figure 15-15.
- **Select Tweak in the Browser.** When you pick a tweak in the browser, the distance or angle of the tweak is shown in an edit box. See Figure 15-16. You can use this edit box to change the distance or angle of the tweak by highlighting the existing value and entering a new value.

Exercise 15-8 Complete the Exercise on the Student CD.

Figure 15-15.
Dragging a component to change the tweak distance.

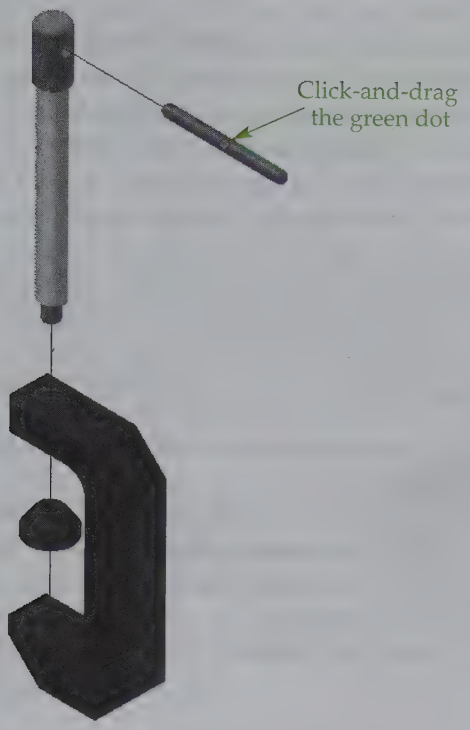
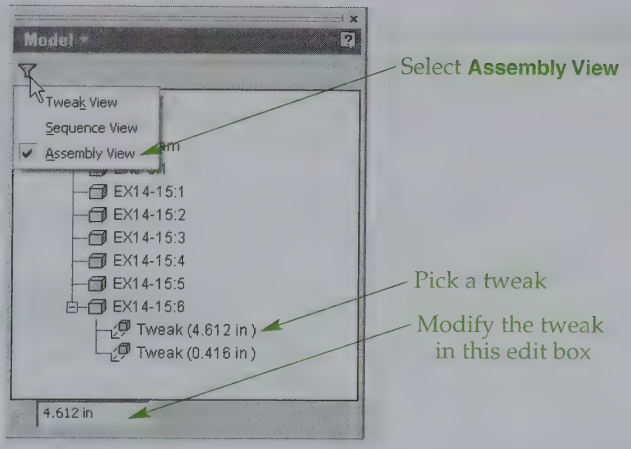


Figure 15-16.
Using the edit box to modify a tweak amount.



Using the Precise View Rotation Tool

The presentation environment contains all the view tools with which you have become familiar, including **Zoom**, **Pan**, **Rotate**, and **Look At**. Any of these tools can be used to set the desired *camera view*, which is the term used to define the position and zoom parameters of the presentation view. However, standard view tools do not allow you to rotate the presentation view using a specified incremental value, such as 10° in a certain direction. This type of view manipulation is managed using the **Precise View Rotation** tool. To access the **Precise View Rotation** tool pick the **Precise View Rotation**

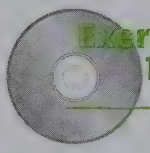
button on the **Presentation Panel** bar; pick the **Precise View Rotation** button on the **Presentation Panel** toolbar; right-click in the graphics window, and select the **Precise View Rotation...** menu option; or select the **Precise View Rotation** option from the **View** pull-down menu.

When you access the **Precise View Rotation** tool, the **Incremental View Rotate** dialog box is displayed. See Figure 15-17. This dialog box functions the same as the **Incremental View Rotate** dialog box, described later in this text, used for creating a custom drawing view. The **Incremental View Rotate** dialog box allows you to precisely rotate the presentation view, using a specified angle of rotation in a certain direction. Use this dialog box by entering the amount, or angle, of rotation in the **Increment** edit box. Then, pick the rotate buttons to rotate the view to a specific position. Each time you select one of the rotate buttons, the view rotates in the specified direction at the defined increment. If you want to return to the original view, pick the **Reset** button. Finally, when you have rotated the view as needed, pick the **OK** button to close the **Incremental View Rotate** dialog box, and exit the **Precise View Rotation** tool.

PROFESSIONAL TIP

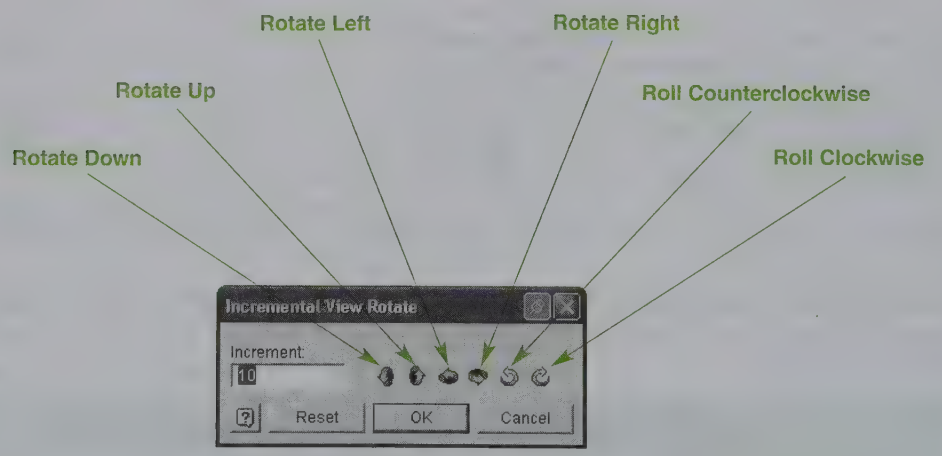


Once you define a specific presentation camera view using the **Precise View Rotation** tool, you may want to use the **Save Camera** option to save the current presentation view display.



Exercise 15-9 Complete the Exercise on the Student CD.

Figure 15-17.
The **Incremental View Rotate** dialog box has buttons to control the direction and an **Increment** edit box to control the amount of view rotation.



Animating Presentations

Once you create and explode a presentation view, you can display the assembly characteristics and tweaked component parameters by animating the presentation. An *animation* is useful for visualizing how assembly components fit together. In addition,

a presentation animation allows you to further display product assembly and disassembly characteristics for multiple purposes, including maintenance and assembly requirements. The **Animate** tool is the principal tool used for animating presentation views, and allows you to fully define, play, and record an exploded presentation view animation. Another animation option is to set the browser filter as **Sequence View** and use the sequence view options to show the animation. However, using the sequence view technique does not allow you to edit the animation and is typically used in conjunction with the **Animate** tool.

Animating Using Browser Sequence Views

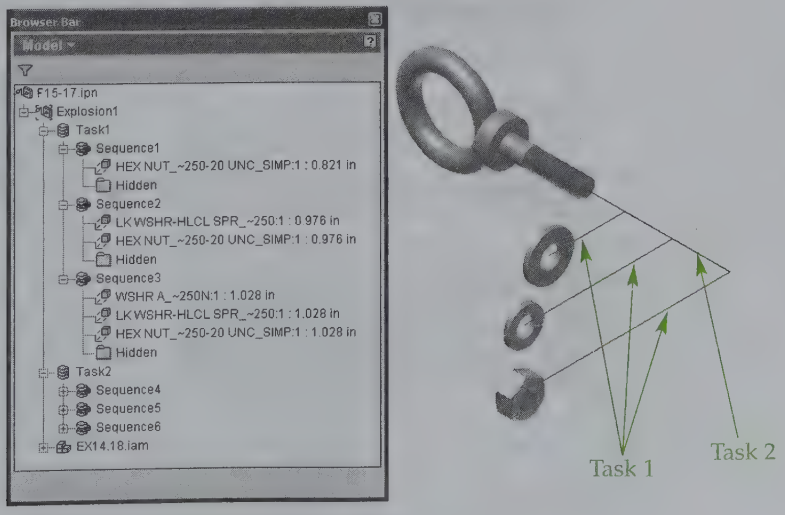
The **Sequence View** browser filter option is typically used to help develop task and sequence specifications, especially for animating presentation views. When the **Sequence View** filter option is specified, presentation view tasks and sequences are displayed. A ***task*** is a group of sequences that define a certain assembly or disassembly relationship. A ***sequence*** is a group of tweaks that identify tweak relationships between components. Tasks describe how groups of components function in the animation, while sequences describe how each component functions in reference to other components during the animation. For example, Figure 15-18 shows a presentation with two tasks. The first task contains three sequences. The first sequence aligns the washer with the bolt shaft, the second sequence aligns the lock washer with the bolt shaft, and the third sequence aligns the nut with the bolt shaft. The second task also contains three sequences, which assemble the washer, lock washer, and nut, with the threaded eyebolt shaft as specified with tweaks.

NOTE



Although the presentation in Figure 15-18 does contain two tasks, all the sequences from both tasks can be placed in one task. As shown in the browser display of Figure 15-18, the sequences are not really separate, as evidenced by the sequence numbers 1-6. Typically, multiple tasks are used for larger assembly presentations, where distinct components or subassemblies are animated.

Figure 15-18.
An example of a presentation with two tasks.

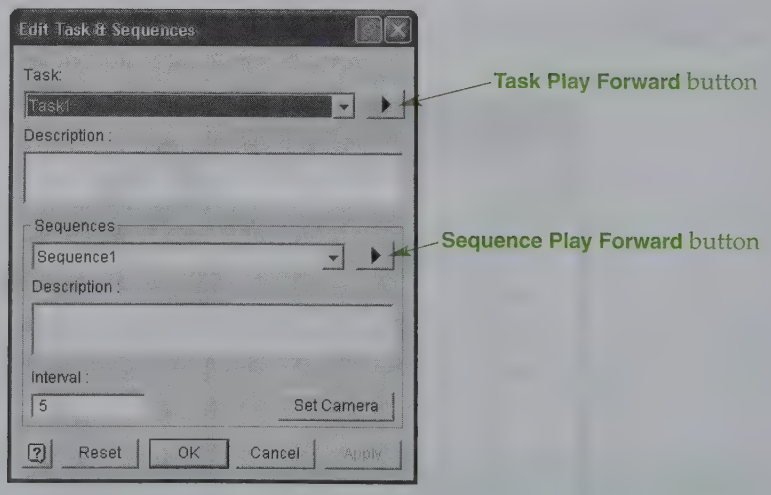


Task and sequencing of presentation view animations can be complicated and difficult to understand, especially for large, complex presentation views. The key to animation is to effectively manage tweaks, tasks, and sequences. To view the effects of a task or sequence, right-click on the task or sequence in the browser, and select the **Edit...** menu option. When you edit a task or sequence, the **Edit Task & Sequences** dialog box is displayed, with the specified task or sequence active. See Figure 15-19. If you edit a specific task, the task is active and displayed in the **Task** drop-down list. Similarly, if you edit a sequence, the sequence is active and displayed in the **Sequence** drop-down list. Still, you can choose to edit a different task or sequence by picking a task or sequence from the **Task** or **Sequence** drop-down lists. Then you can enter a description of the animation task or sequence in the corresponding **Description** area. Descriptions are typically used for clarification and future task and sequence editing reference. To observe the specified task or sequence operation, pick either the task or sequence **Play Forward** button. After you play the animation, pick the **Reset** button to return to the exploded view.

In addition to observing the animation operation of certain tasks or sequences and defining task and sequence descriptions, use the **Edit Task & Sequences** dialog box to specify the sequence interval and sequence camera view. The sequence interval is the animation playing speed. To create a faster playing speed, enter a higher interval value in the **Interval** edit box. To create a slower playing speed, enter a lower interval value in the **Interval** edit box.

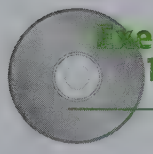
Although you may set a camera view for the entire presentation view, you can also create a separate sequence camera view. When you define a sequence camera view, then animate the presentation, the animated display automatically zooms in or out, and rotates according to the specified camera view. To identify a sequence camera view, first rotate and zoom the presentation view as desired using viewing tools. Then pick the **Set Camera** button to create the sequence camera view. Once you specify all the **Edit Task & Sequences** dialog box options, pick the **Apply** button to apply the descriptions and camera setting. Then continue working with the dialog box, or pick the **OK** button to exit the tool.

Figure 15-19.
The **Edit Task & Sequences** dialog box.



In addition to accessing the **Edit Task & Sequences** dialog box from the sequence view presentation browser, you can also accomplish the following processes:

- **Create a new task.** To create a new task, right-click on the presentation view in the browser, such as Explosion1, and select the **Create Task** menu option. Once the new task is present, you can add sequences by creating additional tweaks or drag existing sequences into the new task.
- **Create a new sequence.** Sequences contain tweaks. If a sequence contains more than one tweak, you can create a new sequence by right-clicking on a tweak and selecting the **New Sequence** menu option. The new sequence becomes the first sequence of the first task, and contains the specified tweak. As a result, the tweak is removed from the original sequence.
- **Modify sequence order.** Modifying sequence order is one of the most useful purposes of the **Sequence View** browser filter option, and allows you to redefine animation operations. Changing sequence order is accomplished much the same way as dragging a sequence into a different task. To edit sequencing, drag the sequence above or below another sequence, depending on the requirement. For example, if a washer is set as Sequence1 and a lock washer is set as Sequence2, but you want the lock washer to assemble first, drag Sequence2 above Sequence1. In this example, Sequence2 will automatically become Sequence1.
- **Group sequences.** You can group sequences to add tweaks of two or more sequences together. This operation is done by holding down the [Ctrl] key and selecting each of the sequences you want to group. Then, right-click and select the **Group Sequence** menu option.



Exercise 15-10 Complete the Exercise on the Student CD.

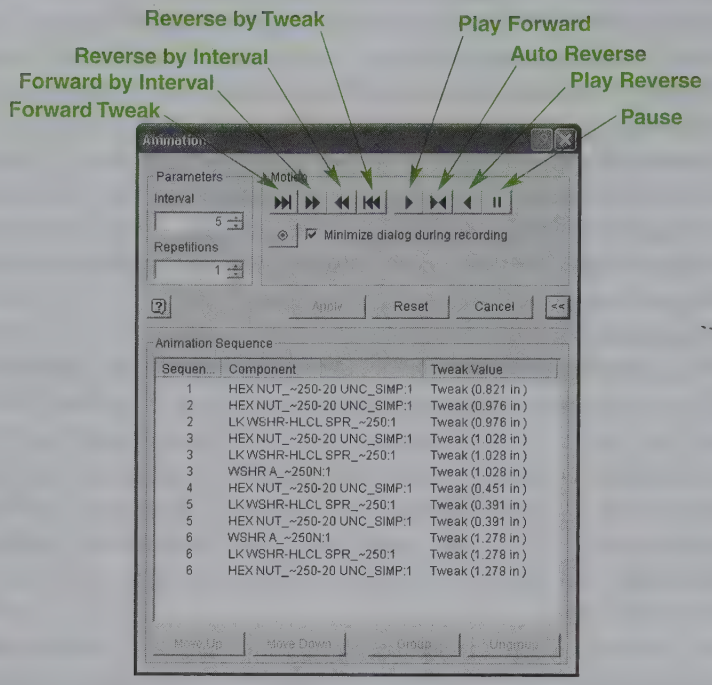
Animating Using the Animate Tool

Once you fully create a presentation view, you can use the **Animate** tool to define animation characteristics, play the animation, and record an exploded presentation view animation, using specified task and sequencing information. To access the **Animate** tool, pick the **Animate** button on the **Presentation Panel** bar; pick the **Animate** button on the **Presentation Panel** toolbar; right-click in the graphics window, and select the **Animate...** menu option; or select the **Animate...** option from the **Tools** pull-down menu.

When you access the **Animate** tool, the **Animation** dialog box is displayed. See Figure 15-20. Use the following areas in the **Animation** dialog box to set presentation view animation:

- **Parameters.** This area is used to define the interval and repetition characteristics of the animation. The ***animation interval*** functions similar to the sequence interval described earlier and specifies the animation playing speed. The interval identifies how many steps are required to move a component a specified distance. For example, if a tweak distance is 5", and you define an interval of 5, each step moves the component 1". To create a faster playing speed, enter or select a higher interval value in the **Interval** edit box. To create a slower playing speed, enter or select a lower interval value in the **Interval** edit box. The ***animation repetition*** is the number of times the animation goes through the animation process. The default repetition of 1 allows you to view the assembly or disassembly animation one time. Enter or select the desired number of repetitions from the **Repetitions** edit box.

Figure 15-20.
The **Animation** dialog box.



NOTE



After you modify the interval and repetition specification, you must pick the **Apply** button before you can set the animation in motion.

- **Motion.** This area allows you to play and record the animation. The following options are available in the **Motion** area:
  - **Forward By Tweak.** Pick this button to move the animation forward by one tweak.
  - **Forward By Interval**. Select this button to advance the animation forward by one interval step. Using the previous example of a 5" tweak distance and an interval of 5, the component will move forward 1".
  - **Reverse By Interval.** Use this button to move the animation back by one interval step. You can only move an animation in reverse if it has been advanced forward.
  - **Reverse By Tweak.** Select this button to reverse the animation by one tweak. Again, you can only move an animation in reverse if it has been advanced forward.
  - **Play Forward.** Pick this button to play the entire animation forward.
  - **Auto Reverse.** Select this button to play the animation forward, then once the forward cycle ends, the animation automatically plays in reverse.
  - **Play Reverse.** Pick this button to play the entire animation in reverse. If you have not advanced the animation forward, when you select the **Play Reverse** button, the presentation will begin as the assembled product, and then disassemble.
  - **Pause.** Select this button to pause the animation.
  - **Record.** Pick this button to record the presentation view animation as an animation file. Before you select the **Record** button, you may want to determine whether or not you want the **Animation** dialog box to be displayed in the recording. If you do not want the **Animation** dialog box shown, select

the **Minimize dialog during recording** checkbox. When you select the **Record** button, the **Save As** dialog box appears, allowing you to specify a location for the animation file. You can also choose the file format by selecting either **AVI** or **WMV** from the **Save as type:** drop-down list.

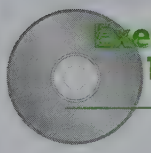
- **Animation Sequence.** This area is available when you select the **More** button, and is used to set the animation sequence, much like the sequence view browser options previously described. The **Animation Sequence** area contains **Sequence**, **Component**, and **Tweak Value** lists. The sequence numbers represent the order in which the components and corresponding tweaks are animated. Components that have the same sequence number are grouped as tasks and move together during the animation sequence. You can remove a sequence from a task by selecting the sequence and picking the **Ungroup** button. Conversely, you can add a sequence or a number of sequences together to form a task, or add sequences to an existing task. To apply this technique, hold down the [Ctrl] key and select each of the sequences you want to group together. Then, pick the **Group** button to generate the task.

When working with the **Animation** dialog box, pick the **Reset** button at any time to reset the animation and return the presentation view to its original display. Once you finish the animation session, pick the **Cancel** button to exit the **Animate** tool.

NOTE



When you pick a sequence in the **Animation** dialog box, the tweak trail is highlighted in the graphics window. After you modify the animation sequence specifications, you must pick the **Apply** button before you can set the animation in motion.



Exercise 15-11 Complete the Exercise on the Student CD.

Placing Additional Presentation Views

A presentation file can contain as many presentation views as needed. For some presentations only one presentation view is required, while other presentations use multiple presentation views. Although there are numerous reasons for having more than one presentation view, typically additional presentation views allow you to display different views of the assembly and create different types of animations. For example, you can show an assembly with different component colors, or with some components suppressed. You can also display an animation of an alternative product assembly technique.

Creating additional presentation views is very similar to placing the initial presentation view using the **Create View** tool, previously described. When you access the **Create View** tool, the **Select Assembly** dialog box is displayed. However, the **File** drop-down list and file **Browse** button are not available, because the assembly file used for the presentation is already defined when the initial presentation view is created. You cannot create presentation views using different assembly files in one presentation file. To create a presentation view of a different assembly, you must open a new presentation file. To insert additional presentation views, select a design view file from the **Design View** drop-down list, or use the **Browse** button to locate the desired design view file.

A new presentation view is identified as Explosion2 in the browser, assuming only one other presentation view already exists. For example, if there are three existing presentation views, the new view is named Explosion4. As with other Inventor applications, you may find it helpful to rename presentation views by picking and highlighting the name and entering a new name. Once created, additional presentation views can be manipulated as required, using the tools identified in this chapter and previous chapters. The active presentation view is displayed in the graphics window, while the inactive presentation views are shaded in the browser. To activate a different presentation view, right-click on an inactivated view and select the **Activate** menu option, or double-click on the view name in the browser.

NOTE

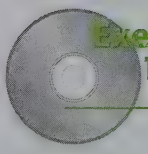


Assembly design views allow you to define the display of a presentation view and can be very useful for creating additional presentation views.

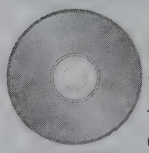
PROFESSIONAL TIP



Right-click on an item in the browser or a component in the graphics window and select the **Restore Camera** option to return the presentation view display to the previously saved camera view. Pick the **Save Camera** option to set the specified presentation view display as the default view, which allows you to identify the desired position and zoomed display of the presentation view. You can then return to the saved view if modifications are made, such as rotating the presentation view or zooming in or out.



Exercise 15-12 Complete the Exercise on the Student CD.



Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. What is a *presentation*?
2. Identify the Inventor files that can be used for presentations, and indicate why these are the only files that can be used.
3. What does it mean when it is said that presentations are essentially exploded assemblies?
4. Identify at least two uses for presentations.
5. Define *trails*.
6. Briefly define the word *exploding*.
7. Give another name for exploding an assembly in a presentation view and provide a useful application.
8. Describe *tweaks* and briefly discuss the difference between creating tweaks automatically and manually.
9. Briefly explain the function of the **Tweak Components** tool.

10. Define the term *camera view.*
11. Give two uses for an animation.
12. Define *task.*
13. Provide a definition of *sequence* and explain the difference between task and sequence.
14. Briefly discuss the function of the **Animate** tool.
15. Explain the function of the animation interval and give an example.

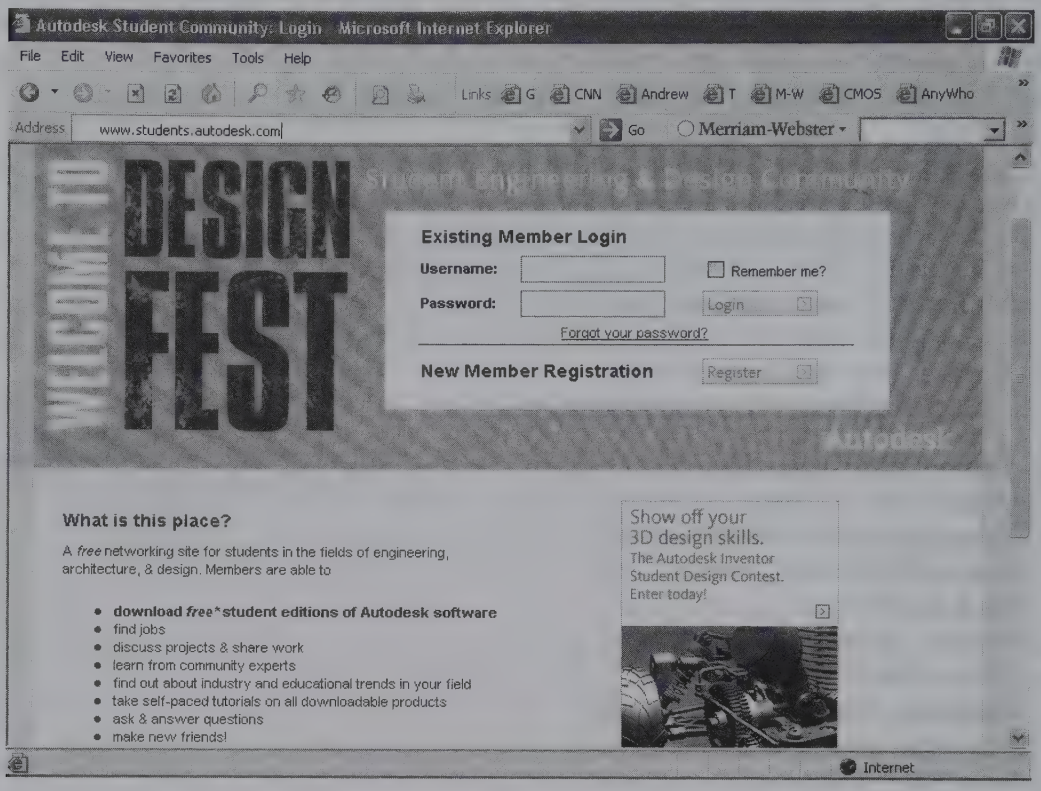
Problems

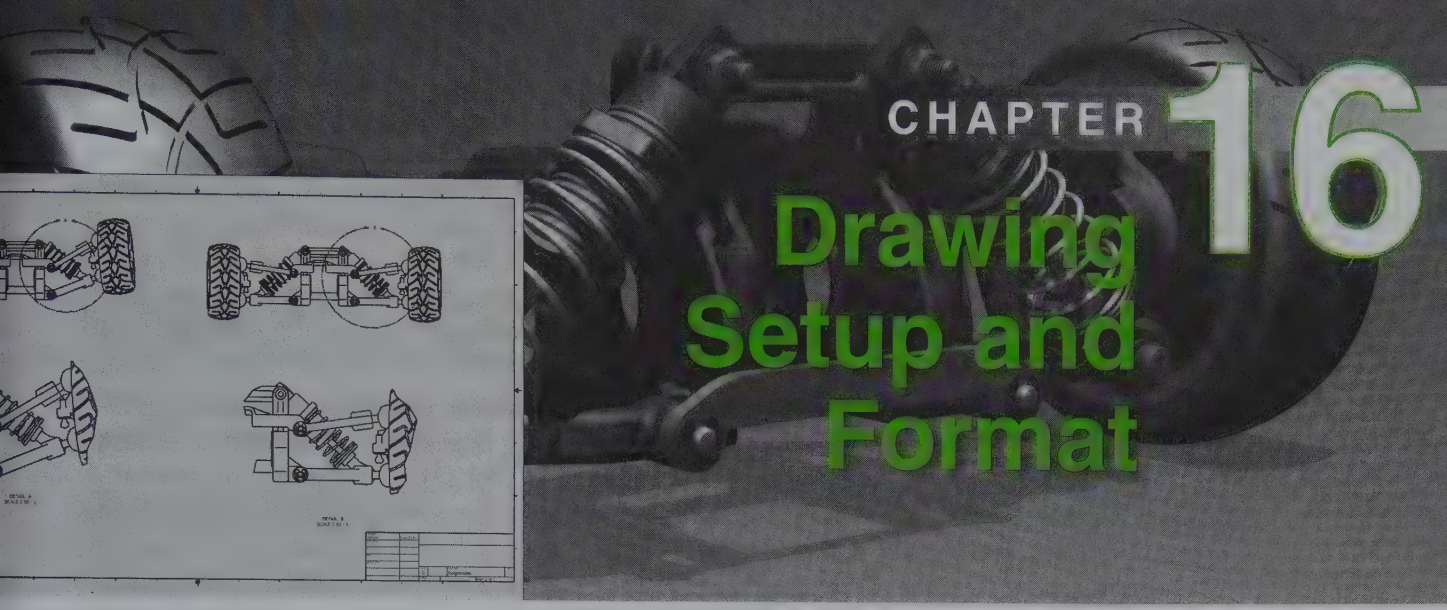
Instructions:

- Begin a new presentation file for each of the projects.
- Create a presentation view using the specified file.
- Use the information provided in this chapter to develop presentation files.
- Explode and animate the initial presentation view.
- Create at least one more presentation view.
- Explode, manipulate, and animate the additional presentation views so they differ from the initial view.
- Save the file using the specified name.

1. **Name:** Bottle Assembly
   Units: Inch
   Template: PRESENTATION-in.ipn
   Assembly File: P12-1.iam
   Save As: P15-1.ipn
2. **Name:** Pipe Assembly
   Units: Metric
   Template: PRESENTATION-mm.ipn
   Assembly File: P12-2.iam
   Save As: P15-2.ipn
3. **Name:** Rotating Hinge Assembly
   Units: Inch or Metric
   Template: PRESENTATION-in.ipn or PRESENTATION-mm.ipn
   Assembly File: P12-3.iam
   Save As: P15-3.ipn
4. **Name:** Cam Assembly
   Units: Inch
   Template: PRESENTATION-in.ipn
   Assembly File: P12-4.iam
   Save As: P15-4.ipn
5. **Name:** Fork
   Units: Metric
   Template: PRESENTATION-mm.ipn
   Assembly File: P12-5.iam
   Save As: P15-5.ipn
6. **Name:** Housing Assembly
   Units: Inch
   Template: PRESENTATION-in.ipn
   Assembly File: P12-6.iam
   Save As: P15-6.ipn
7. **Name:** Push Pin
   Units: Inch
   Template: PRESENTATION-in.ipn
   Assembly File: P12-7.iam
   Save As: P15-7.ipn

The home page of the Autodesk Student Engineering and Design Community Web site is www.student.autodesk.com. This Web site offers valuable information about industrial and education trends in the fields of manufacturing and product design. This site allows students to network with other students and community experts. Students can also download free student editions of Autodesk software, take self-paced tutorials on the software, and search for jobs.





CHAPTER 16

Drawing Setup and Format

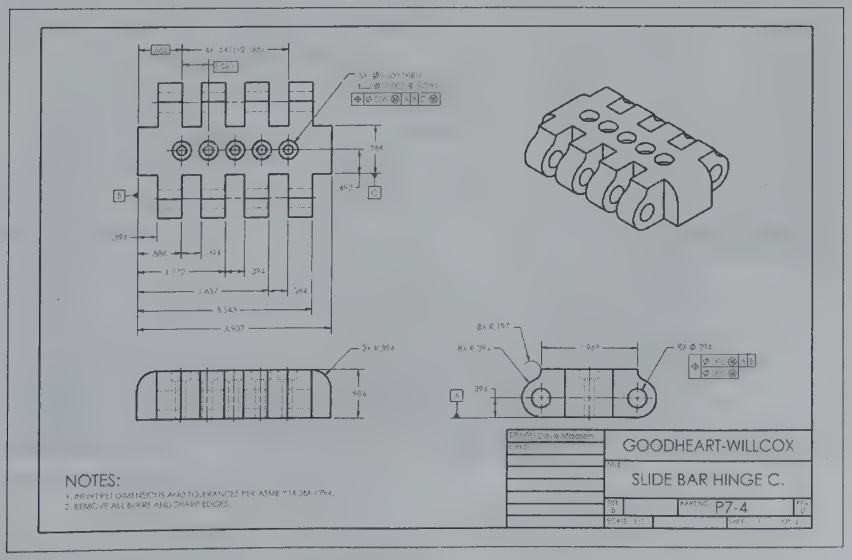
Learning Objectives

After completing this chapter, you will be able to do the following:

- Work with drawing styles.
- Use layers.
- Work with object defaults.
- Use drawing standards.
- Adjust drawing document settings.
- Set drawing application options.
- Specify drawing sheet parameters.
- Use default borders and create a custom border.
- Use default title blocks and create a custom title block.

Inventor combines three-dimensional (3D) solid modeling with two-dimensional (2D) drawing capabilities. An example of a 2D drawing created by referencing a 3D model is shown in Figure 16-1. Part, assembly, and weldment drawings are created

Figure 16-1.
An example of a two-dimensional drawing created by referencing a three-dimensional model.



quickly and easily from existing part models. Additionally, due to the parametric nature of Inventor, when you edit a model, the corresponding drawing modifies to reflect the new design. You can also edit a model by modifying the parametric model dimensions inside a drawing.

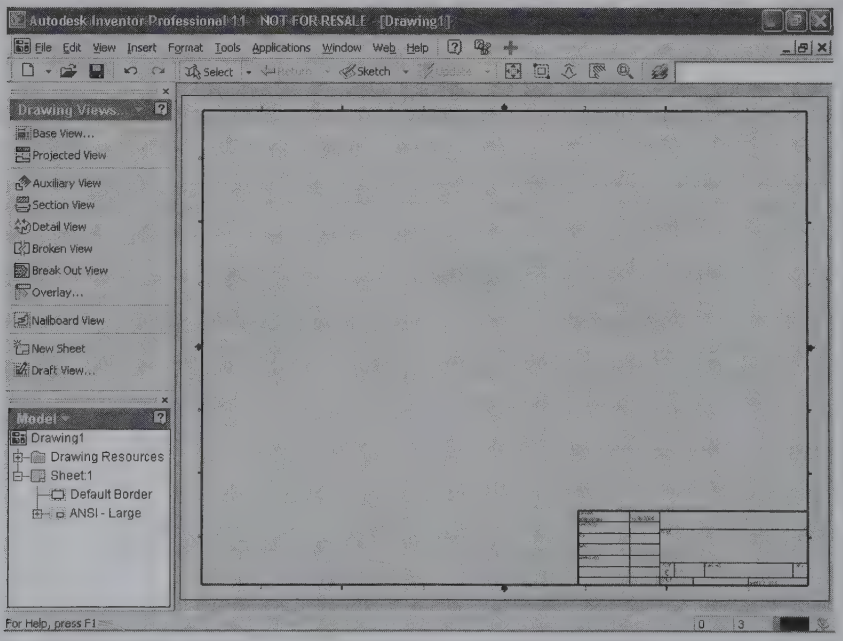
Developing Drawings

Inventor drawing files are separate files that carry the extension .idw. You can create a variety of fully detailed drawings using existing model geometry and additional drawing tools. There are a number of methods for creating drawings using Inventor. For example, you may want to initially place drawing views, followed by adding annotations, and finally define a border and a title block. Or, you may want to place drawing views, followed by specifying a border and title block, and then add annotations toward the end of the drawing process. Any of these techniques can be effective. However, this book approaches the creation of drawings in the following order:

1. Open a template containing predefined iProperties, drafting standards, document settings, sheet parameters, a border and a title block.
2. Adjust iProperties, drafting standards, document settings, sheet parameters, and border and title block characteristics according to individual drawing requirements.
3. Place and create drawing views.
4. Add drawing annotations and detail information.

By default, when you open a new Inventor drawing file, the work environment should look like the display in **Figure 16-2.** A default sheet, border, and title block are automatically inserted into the drawing. However, you have full control over the drawing display, and can modify drawing information as required.

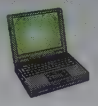
Figure 16-2.
The default Inventor drawing work environment.



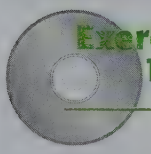
A drawing ***sheet*** is the base of a drawing and represents the physical limits of the drawing area, or the paper size. Think of the sheet as the piece of paper on which you are drawing. Drawing sheets contain all the drawing information and items, including a border, a title block, and one or more drawing views. In addition, special symbols, dimensions, and all other annotations displayed on the drawing are located on sheets.

By default, there is only one sheet in a drawing file. If you are working with a single part, for example, you may need only one sheet. However, when you generate more than one part, each of which is a separate assembly component, you may choose to create several sheets in a single drawing file. For example, a drawing file may contain a sheet with an assembly drawing, and a number of sheets that contain the individual part details.

PROFESSIONAL TIP



A drawing template that contains predefined settings and options according to specific industry and company standards should be used to create drawings.



Exercise 16-1 Complete the Exercise on the Student CD.



Exercise 16-2 Complete the Exercise on the Student CD.

Styles and Standards

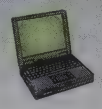
Inventor styles are used in the model work environment to define specific color, lighting, and material characteristics used when creating 3D solid models. Styles are also important when creating 2D drawings. Drawing styles allow you to control drawing display characteristics including dimension and annotation style, and multiple other drafting elements. Many drawing characteristics are defined using styles. Using drawing styles can be confusing, but just remember the following:

- There are several style types, each associated with a specific item. For example, the dimension style describes the display of dimensions, while the center mark style defines the display of center marks.
- A single style may contain many variables. For example, a dimension style describes numerous dimension characteristics including dimension text height and arrowhead style.
- Many styles reference other styles for certain information. For example, the dimension style references the text style for dimension text characteristics.
- Every drawing object is associated with a particular object default that identifies what styles and layers are available or used when creating the drawing.
- A standard contains selected styles from each style type. Drawing styles associated with the active standard are applied to the drawing when objects and annotations are added.

- There are numerous default styles and standards available with Inventor. If a default standard is appropriate for your application, it is just a matter of activating the standard. If a default standard is not acceptable, you may be able to adjust the standard by selecting different default styles.
- If none of the default styles or standards work for your application, create a copy of an existing style or standard that is similar to your expectations, edit the new style or standard as needed, and then save the style to the styles library, so that can be used for future models.

The following content introduces you to Inventor default styles for the most commonly used drafting and design applications. The introduction to each style type is followed by an exercise or exercises that allow you to create a modified or custom style for the style type. These exercises give you the opportunity to create styles that comply with the American Society of Mechanical Engineers (ASME) national standards using metric units. Completing the exercises and applying the results as instructed allows you to save the styles to your template for future use. The end-of-chapter problem allows you to continue creating styles for a customary inch template that complies with ASME standards. The most commonly used drafting applications, such as layers, text, and dimension styles, are described in the textbook. Applications, such as geometric tolerancing and welding symbols, that are common but may not be used by every company or school, are described in the CD that accompanies this textbook.

PROFESSIONAL TIP

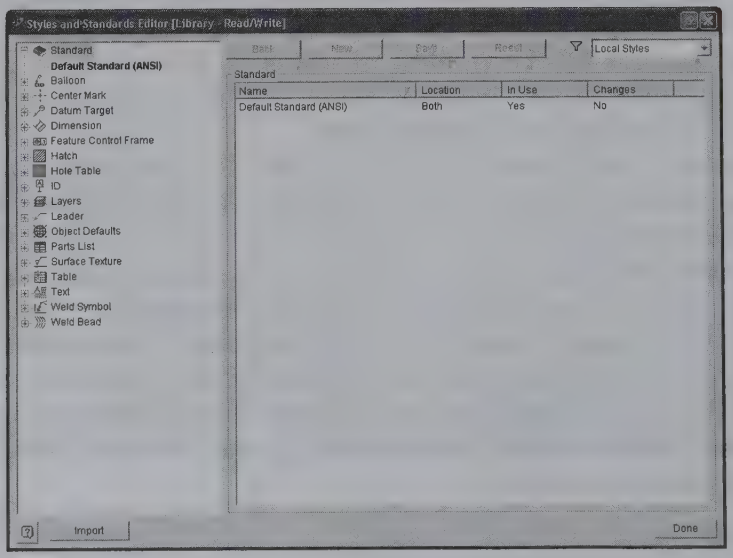


Styles can be created and modified at any time during the design process. However, setting drawing styles in your templates is very effective because the same display characteristics are automatically applied each time you develop a certain type of drawing.

The Styles and Standards Editor

The Inventor **Styles and Standards Editor**, shown in Figure 16-3, is the primary resource for observing and modifying the characteristics of existing drawing styles

Figure 16-3.
The **Styles and Standards Editor, Standard** area shown.



and standards. You can also use the **Styles and Standards Editor** to create new styles and standards and import styles and standards. Access the **Styles and Standards Editor** by selecting the **Styles Editor...** option from the **Format** pull-down menu.

The **Styles and Standards Editor** contains a tree structure, or browser, a style or standards specific area with multiple options, a filter drop-down list, and a number of buttons. Inside the browser are multiple parent nodes associated with style types or standards. Before you explore the style or standard types, you may want to become familiar with the following general items found in the **Styles and Standards Editor**. Each of these options functions the same regardless of the style type or standard with which it is working:

- **Back.** Pick this button to return to the previous **Styles and Standards Editor** display.
- **New.** Select this button to create a copy of the selected style or standard. When you pick the **New** button, the **New Style Name** dialog box is displayed. See Figure 16-4. Use the **New Style Name** dialog box to create a new style or standard, by entering a name for the new style or standard in the **Name** edit box, and picking the **OK** button to create the new style or standard. When creating a new style, the **Add to Standard** check box is available, which if selected adds the new style to the list of available styles for the active standard. When creating a new standard, the **Based On:** drop-down list is displayed, and is used to select the standard from which you want to base your standard. A new style or standard is a copy of the existing style or standard you selected, and is usually modified as needed.
- **Save** and **Reset**. If you edit style or standard settings, you must save the changes by picking the **Save** button. If you do not save the changes, you will be prompted to do so before continuing. You also have the option of returning style or standard changes back to the default settings by selecting the **Reset** button before saving changes.

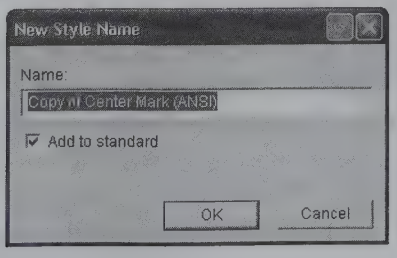
NOTE



Saving changes using the **Save** button only saves the changes in the current file, and does not save changes to the style library.

- **Filter.** Use this drop-down list to display only certain styles and standards in the **Styles and Standards Editor**. You can choose to list all the styles and standards available in the styles library and those used and created in the file by selecting the **All Styles** option, or you can show only styles and standards used, active, and created in the file by picking the **Local Styles** option. Another option is to display only styles and standards associated with the active standard by picking the **Active Standard** option. The **Local Styles** and **Active Standard** filters

Figure 16-4.
The **New Style Name** dialog box.



may be helpful if you know exactly which styles and standards should be available in the drawing, and you do not want to work around all of the styles and standards found in the styles library.

- **Import.** Pick this button to access the **Import style definition** dialog box, shown in Figure 16-5. The **Import style definition** dialog box functions much like an **Open** dialog box and allows you to import style and standard definition files that carry the extension .styxml.
- **Done.** Select this button when you are finished using the **Styles and Standards Editor**. You can also accomplish this task by picking the **Close** button in the upper-right corner.

When you select one of the parent nodes in the **Styles and Standards Editor** browser, specific content areas are displayed, such as the **Standard** area shown in Figure 16-3. Though the content is specific to the selected style type or standard, the format of each area is the same, and includes the following lists:

- **Name** indicates the name of the style or standard.
- **Location** describes where the style or standard is found in reference to the current file and the style library. Styles and standards found in the style library are identified by **Library**, styles and standards located in the current file that have not been saved to the style library are identified as **Local**, and styles and standards that are found in the style library but are also used or active in the current file are identified as **Both**.
- **In Use.** This list identifies whether or not a style or standard is used in the current file. If a style or standard is active or is used in the file, the **In Use** description is **Yes**.
- **Changes** describe the style or standard status. Initially all styles and standards are listed with **No** changes. When you edit a style or standard but do not save the changes to the style library, **Local** is shown. If you edit a style or standard and save the changes to the style library, **Library** is shown. If a style or standard has been edited in the current file, and the style library is modified, resulting in different values in both places, **Both** is displayed.

When you right-click on a style type or standard child node in the browser, or the style or standard name located in the **Name** list, the following pop-up menu options are available, as shown in Figure 16-6:

Figure 16-5.
The **Import style definition** dialog box.

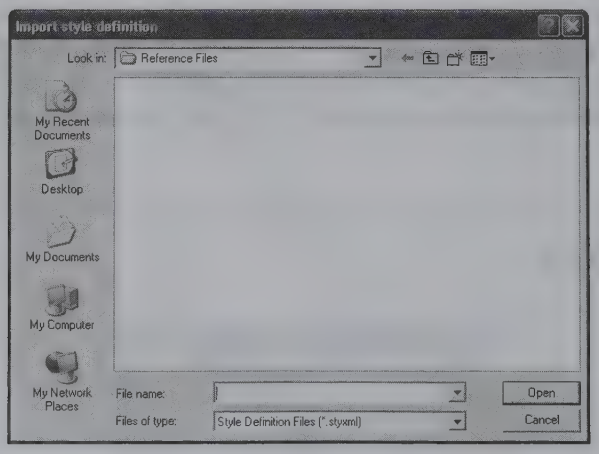
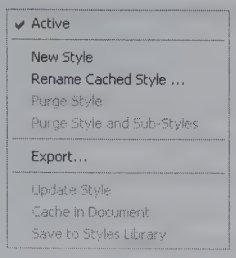


Figure 16-6.
The style pop-up menu.



- **Active.** This option is only available when you right-click on a standard, and activates the selected standard for use in the drawing. An active standard is the standard that is currently used, or turned on, in the drawing, and is displayed as bold in the **Styles and Standards Editor**.
- **New Style.** This option functions the same as selecting the **New** button.
- **Rename Cached Style.** Pick this option to access the **New Style Name** dialog box. You can rename a style or standard using the **New Style Name** dialog box if the style or standard has not been saved to the style library, but when you try to rename a style or standard that is contained in the style library, a copy of the style or standard is created, much like using the **New** or **New Style** options.
- **Replace Style.** This option is only available when you right-click on a style, and activates the **Replace Style** dialog box. Use the **Replace Style** dialog box to use a different style of the same type, for example balloon styles, in-place of the selected style. Pick the replacement style from the **With** drop-down list, and choose the **Purge replaced style** check box to remove the original style from the file.
- **Purge Style.** Pick this option to remove an unused style or standard from a file. If a style or standard has not been saved to the library, the style or standard will be deleted when purged. However, if a style or standard has been saved to the styles library, the style or standard will be removed from the current file, but is still available in the style library.
- **Purge Style and Sub-Styles.** This option is only available when a style or standard references a custom sub-style, and allows you to purge all styles or standards associated with the selection.
- **Export...** Pick this option to export the selected style or standard as a style definition file (*.styxml).
- **Update Style.** Select this option to update the style or standard according to the style or standard saved in the style library. Updating a style or standard is done anytime the style library is modified, and you want to update the styles and standards found in a file according to the changes.

NOTE



If you edit a style or standard, but do not save the changes to the style library, picking the **Update Style** option removes any modification you made to the style or standard.

- **Cache in Document.** Styles and standards are located in the styles library, but can be added to files at any time. This situation is apparent when you pick the **Local Styles**, **Filter** drop-down list option, and notice that by default only styles and standards available in the specific drawing are found, or cached in the current document. Pick the **Cache in Document** option to add a style or standard from the styles library to the current file.

- **Save to Styles Library.** Select this option to save a style or standard to the styles library. In order for new styles and standards and modifications made to existing styles and standards to be used in other files, you must save the changes to the styles library.

NOTE



You cannot update or save styles or standards to the styles library if the active project **Use Styles Library definition** is set to No. You cannot save styles or standards to the styles library if the active project **Use Styles Library definition** is set to **Read Only**.

PROFESSIONAL TIP



Use caution when editing and saving styles and standards to the styles library. If you think you may want to access default styles and standards in the future, it is always best to create a copy of a style or standard, edit the copy, and then save the new style or standard to the styles library.

Using Layers

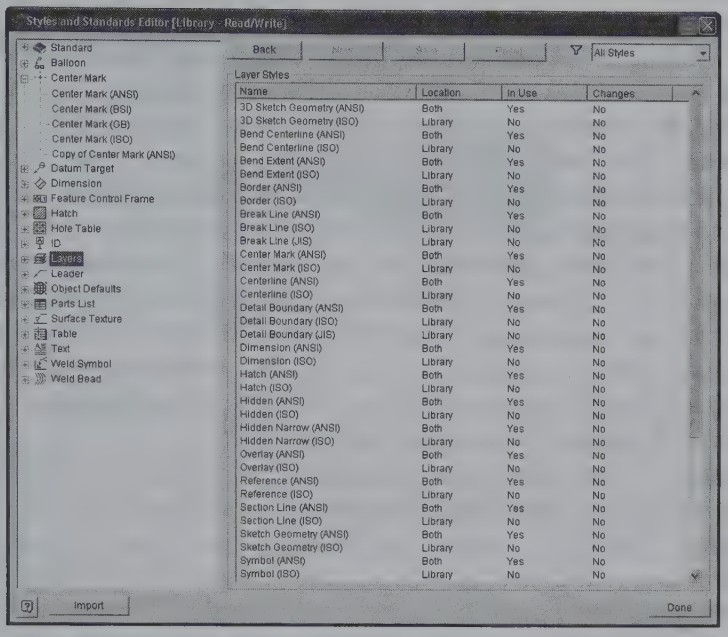
Inventor uses layers in the drawing environment to help differentiate drawing objects. Every item, or group of objects, on a drawing is assigned a specific layer. For example, all visible object lines are drawn on a visible object line layer while hidden lines are drawn on a different layer. Layers contain color, line type, and line weight information. For example, the visible object line layer might be named Visible, use a black color, have a continuous line type, and a line weight of 0.6 mm (.02″), while the hidden line layer might be named Hidden, use a green color, have a hidden line type, and a line weight of 0.3 mm (.01″). Layers have several benefits including the ability to improve drawing clarity by assigning a different color, line type and line weight to each layer. Layers can also be turned off to show or plot only specific drawing information, or all layers can be turned on to display and plot a complete drawing.

Layers are assigned to drawing objects automatically in most situations, based on the layer assigned to a specific object default, described later in this chapter. For example, by default when you place a border on your drawing the Border layer is automatically applied. Or, when you add a drawing view, the visible object lines are placed on the Visible layer and the hidden lines are placed on the Hidden layer. Still, you do have control over layer characteristics and can override object layers if needed.

Layers are manipulated using the layers style in the **Styles and Standards Editor**. When you pick the **Edit Layers** button on the **Inventor Standard** toolbar, the layers style in the **Styles and Standards Editor** opens. When you expand the **Layer** node in the **Styles and Standards Editor** browser, all the available layer styles are listed as child nodes. Picking a specific layer style allows you to modify the layer characteristics in the **Layer Styles** area, shown in Figure 16-7. The following information describes each of the items found in the **Layer Styles** area:

- **Layer Name.** This edit box identifies the layer name. You can modify a name by highlighting the existing name and typing in a new one.
- **On.** You can choose to display or hide layers in your drawing, which can be effective for working on complex drawings, displaying only certain drawing information, or anytime you do not want to show or plot objects on a specific

Figure 16-7.
The **Layer Styles** area of the **Styles and Standards Editor**.



layer. A lit (yellow) lightbulb means that the layer is turned on, is visible in the drawing, and will be plotted. An unlit (white) lightbulb means that the layer in turned off, is not visible in the drawing and will not be plotted. Turn a layer on or off by selecting the lightbulb.

- **Color.** Pick this button to open the **Color** dialog box. The color dialog box has been described throughout this book, and is used to select a specific layer color.
- **Line Type.** Use this drop-down list to select the line type. For example, visible object lines are drawn using a continuous line type, which is solid, while hidden lines are drawn using a hidden line type, which is dashed.
- **Line Weight.** This drop-down list includes several line weights specified in the **Standard** area of **Styles and Standards Editor**, described later in this chapter. Line weight defines the thickness of the lines used to create drawing objects. You can select any available line weight, or enter a new value depending on the desired line thickness.
- **Scale by Line Weight.** Pick this check box to scale the line type according to the line weight. This means that when you increase the line weight, the space between, and the length of, the line segments for noncontinuous line types increases.

To create a new, layer pick the **Click here to add** button, enter the layer name, and adjust the layer settings as needed. To add the new style to the list of available styles for the active standard, be sure the **Add new layers to stand** check box is selected.



Complete the Exercise on the Student CD.

Leader Styles

A *leader* is a line that connects to the beginning or the end of a note, symbol, or other drawing object with a horizontal shoulder and is drawn at an angle where it is capped

on the end with an arrowhead that touches the desired feature. Leader styles are used to define leader characteristics. Leader styles are considered sub-styles because several other styles reference leader styles. Balloon, datum target, dimension, feature control frame, ID, surface texture, and weld symbol styles all reference leader styles.

When you expand the **Leader** node in the **Styles and Standards Editor** browser, all the available leader styles are listed as child nodes. Picking a specific leader style displays leader style variables, and the following areas, as shown in Figure 16-8:

- **Terminator.** This area allows you to define the style, size, and aspect of the leader line terminator, such as an arrowhead, depending on the application. To define the terminator style, select the desired arrowhead from the **Arrowhead** drop-down list. Then, enter the size and aspect of the arrowhead in the **Size (X)** and **Aspect (Y)** edit boxes. *Size* defines the length of the arrowhead from the point to the end, while *aspect* specifies the width or thickness of the arrowhead in reference to the size. See Figure 16-9. For example, most arrowheads are drawn at a 3:1 ratio. As a result, the size of the arrowhead would be 3 mm, or .125", while the aspect would be 1 mm, or .042".
- **Line Format.** This area is used primarily to override leader line format. In most situations the **Line Type**, **Line Weight**, and **Color** variables should be set to By Layer. This will automatically assign a line type, line weight, and color to the leader style based on the characteristics of the layer defined in the **Object Defaults Style** area for objects with leaders. However, you can choose to override the line format variables by choosing a line type, line weight, and color other than By Layer.
- **Extension Line**. Use this area to specify the extension line offset and extension line overshoot for applications in which a leader is attached to an extension line. Enter a value in the **Extension Line Offset** edit box to define the extension line offset, or gap, between the extension line and the object. Enter a value in the **Overshoot** edit box to specify the length of the extension line extension from the leader line endpoint.
- **All-Around Symbol**. Use this area to specify the size of the weld all-around symbol for weldment drawing applications that require all-around welding. Pick the **Scale to Text Height** check box to make the all-around symbol the same

Figure 16-8.
The **Leader Style** area of the **Styles and Standards Editor**.

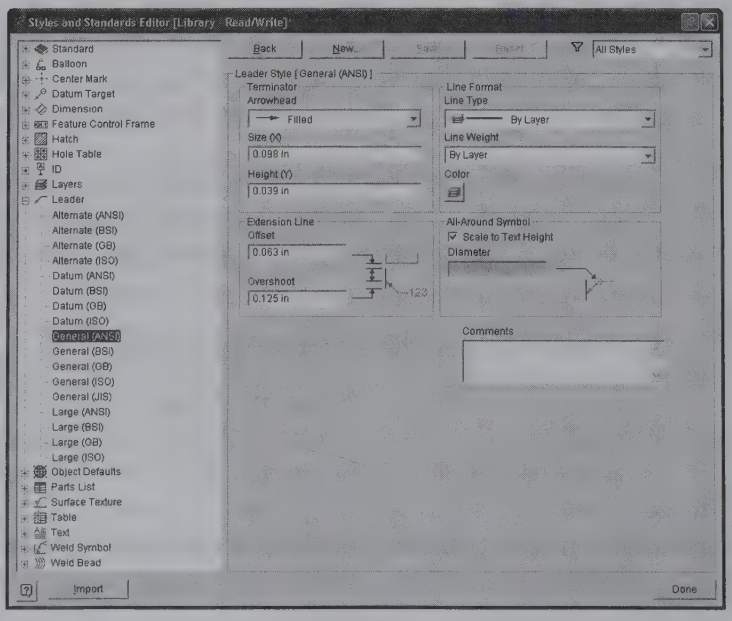
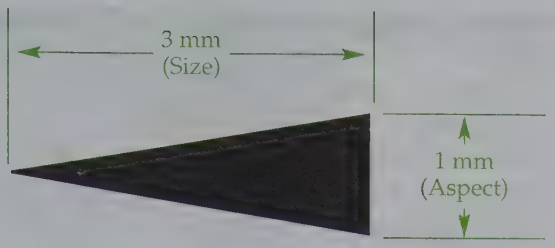


Figure 16-9.
An arrowhead is specified by its size, the aspect is usually 1/3 the size.



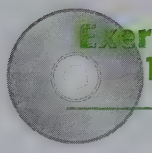
height as annotation text, or deselect the **Scale to Text Height** check box and enter the diameter of the all-around symbol in the **Diameter** edit box.

- **Comments.** Use this edit box to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

PROFESSIONAL TIP



In many cases, the only setting that should change between different leader styles is the terminator type. For example, most leaders require a Filled arrowhead, while datum feature symbols use a Datum 60 Filled arrowhead.



Exercise 16-4 Complete the Exercise on the Student CD.

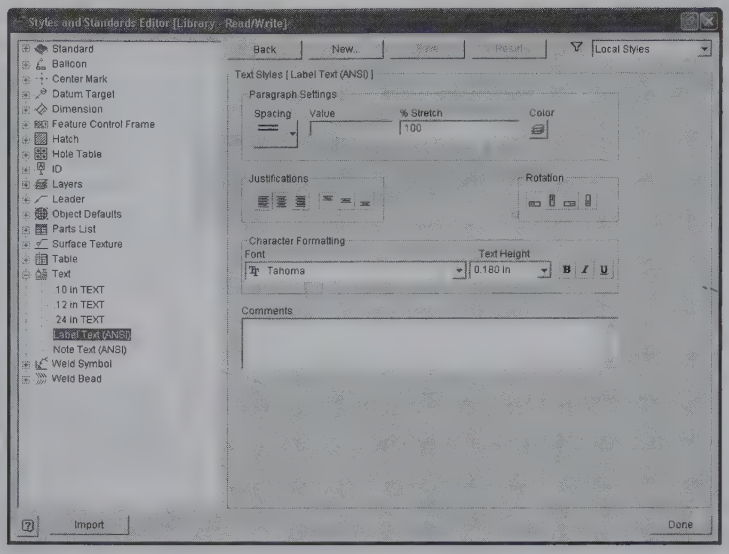
Text Styles

Text styles are used to define text characteristics such as font style and height. Text styles are considered sub-styles because several other styles reference text styles. Balloon, datum target, dimension, feature control frame, hole table, ID, parts list, surface texture, and weld symbol styles all reference text styles.

When you expand the **Text** node in the **Styles and Standards Editor** browser, all the available text styles are listed as child nodes. Picking a specific text style displays text style variables, and the following areas, as shown in Figure 16-10:

- **Paragraph Settings.** This area contains options for adjusting line spacing, text width and color. The **Spacing** flyout button allows you to specify the distance between multiple lines of text. When you select the **Multiple** option, the **Value** edit box becomes useable, allowing you to define a smaller or greater spacing ratio, such as triple or half. When you select the **Exact** option, the **Value** edit box becomes available, allowing you to enter a specific distance between multiple lines of text. The **% Stretch** edit box allows you to define the amount of stretch, or width, of the text. To create a narrower text, enter a percent stretch less than 100, and to create a wider text, enter a percent stretch greater than 100. Pick the **Color**

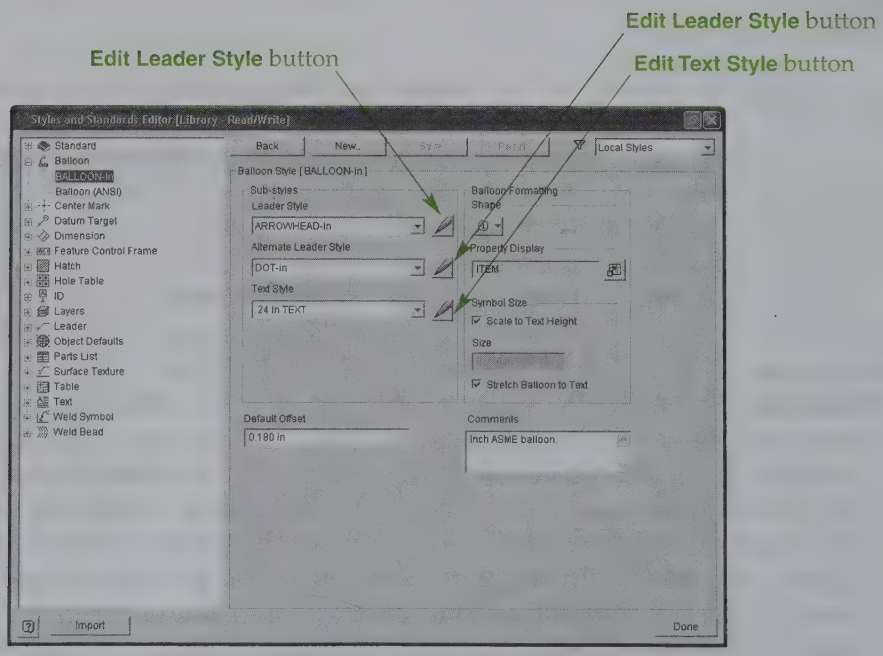
Figure 16-10.
The **Text Styles** area of the **Styles and Standards Editor**.



button to display the **Color** dialog box and define the text color. In most situations **Color** should be set to By Layer, which will automatically assign a text color based on the characteristics of the layer defined in the **Object Defaults Style** area for objects with text. However, you can choose to override the format by choosing a color other than By Layer.

- **Justifications.** The **Justification** buttons allow you to control the left, center, right, top, middle, and bottom justification, or placement, of the text.

Figure 16-11.
The **Balloon Style** area of the **Styles and Standards Editor**.



- **Rotation.** Use the **Rotation** buttons to define the direction of the text. A 0° rotation generates text horizontally from left to right, a 270° rotation generates text vertically from top to bottom, a 180° rotation generates text horizontally from right to left, and a 90° rotation generates text vertically from bottom to top.
- **Character Formatting.** This area controls text font, height and format preferences. Select one of the many font styles from the **Font** drop-down list. RomanS and Century Gothic are common text fonts used in mechanical drafting. The **Text Height** drop-down list allows you to choose or enter a value for the height of the selected text. The text sizes listed are specified in the **Standard** area of **Styles and Standards Editor**, described later in this chapter. You can select any available text size, or enter a new size. Pick one or a combination of the **Bold**, **Italic**, and **Underline** buttons to change the format of the selected text.
- **Comments.** Use this edit box to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

PROFESSIONAL TIP



A typical drawing requires one text style with a height of 3 mm or .12" and a separate text style with a height of 6 mm or .24". These heights are based on ASME text height standards



Exercise 16-5 Complete the Exercise on the Student CD.

Balloon Styles

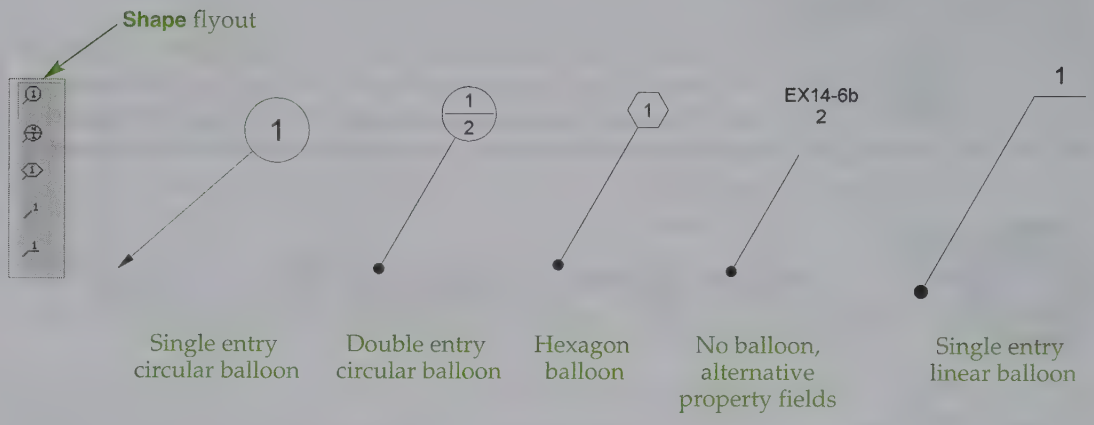
The term *balloon* usually refers to a circle, or other shape, that is connected to an assembly component with a leader. Inside the balloon is an identification number or letter used to define assembly components in relation to a parts list. A *parts list* records and displays the parts and subassemblies used to create the assembly specifications. Balloon styles are used to define balloon characteristics.

When you expand the **Balloon** node in the **Styles and Standards Editor** browser, all the available balloon styles are listed as child nodes. Picking a specific balloon style displays balloon style variables, and the following areas, as shown in Figure 16-11.

- **Sub-styles.** The **Sub-styles** area is used to select existing leader and text styles to use for balloons. Use the **Leader Style** drop-down list to choose the leader style used for most applications. You can access the selected leader style variables by picking the **Edit Leader Style** button. Choosing the **Edit Leader Style** button displays the **Leader Style** area of the **Styles and Standards Editor**. Use the **Alternate Leader Style** drop-down list to choose the leader style used for ballooning application where the leader is dragged from the edge of an object. Usually the only difference between the standard and alternative leader style is the use of a dot terminator style as opposed to a filled arrowhead. As with **Leader Style**, you can access the selected leader style variables by picking the **Edit Leader Style** button. Use the **Text Style** drop-down list to choose the text style used in the balloon. You can access the selected text style variables by picking the **Edit Text Style** button. Choosing the **Edit Text Style** button displays the **Text Style** area of the **Styles and Standards Editor**.

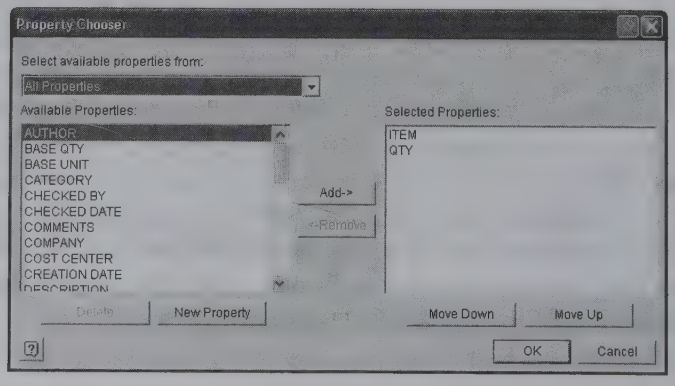
- **Balloon Formatting.** The **Balloon Formatting** area defines the following balloon size and shape preferences:
  - **Shape.** Use this flyout button to select the balloon shape. See Figure 16-12. Pick the **Circular - 1 Entry** button to create a balloon with a single identification number inside a circle. A single entry circular balloon is probably the most common type of balloon and identifies the corresponding item number in the parts list. Select the **Circular - 2 Entries** button to create a balloon with two entries inside a circle. The top number in the balloon represents the item number, while the bottom number specifies the quantity of items. Select the **Hexagon** button to create a balloon with a single identification number inside a hexagon. Choose **None** if you do not want a balloon shape, only a reference number or letter. Pick **Linear** to create a balloon with a single identification number above a leader shoulder.

Figure 16-12.
The **Shape** flyout of the **Balloon Formatting** area is where you can select these shapes.



- **Property Display.** This area contains a display box that identifies the property used to define the balloon value. You can select the properties to be used by picking the **Property Chooser** button to display the **Property Chooser** dialog box. See Figure 16-13. This dialog box allows you to reference properties used in the parts list for component identification purposes. Select the property fields to which you want to have access using the **Select available properties from** drop-down list. The types of properties you select

Figure 16-13.
Select one or more properties and pick the **Add** or **Remove** button.



are shown in the **Available Properties** list box. If you pick the **All Properties** option, all types of properties are shown in the **Available Properties** list box. To create a custom property, pick the **New Property** button, which opens the **Define New Property** dialog box. See Figure 16-14. Pick the **Click here to add new property** button, and enter the new property name. Next, right-click or press [Enter] to accept the property. Continue adding custom properties as needed and select or deselect the **All Properties** check box to activate or deactivate all custom properties. When finished, pick the **OK** button to add the custom properties to the **Selected Properties** list.

To specify which property or properties you want to use in the balloon, choose the desired property from the **Available Properties** list box and select the **Add** button to add the property to the **Selected Properties** list box. When two or more properties are specified, the properties are displayed in the balloon in the same order shown in the **Selected Properties** list box. To move a property up in the list, highlight the property and pick the **Move Up** button. Conversely, to move a property down in the list, highlight the property and select the **Move Down** button. Once you finish specifying property information in the **Property Chooser** dialog box, pick the **OK** button to define the property field balloon style.

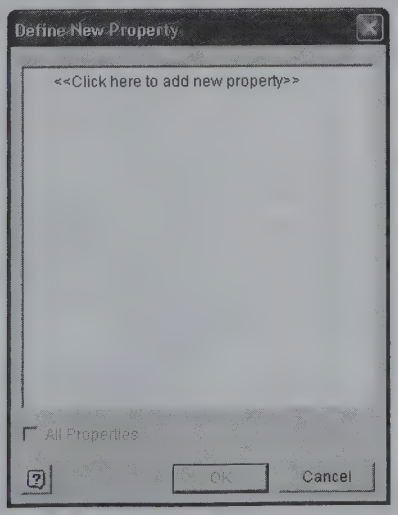
NOTE



In order for the properties you specify in the **Selected Properties** list box of the **Parts List Column Chooser** dialog box to be displayed in the drawing, the properties must also be listed in the parts list.

- **Symbol Size.** This area is only available when you select a **Circular - 1 Entry**, **Circular - 2 Entries**, or **Hexagon** balloon symbol style, and is used to define the size of the balloon symbol. You can choose to size the symbol according to the height of the text by selecting the **Scale to Text Height** check box. If you do not select the **Scale to Text Height** check box, the **Size** edit box becomes available allowing you to enter the exact size, or diameter, of the symbol. A **Stretch Balloon to Text** check box is also available, which allows the balloon to adjust its size according to the width of the text.

Figure 16-14.
The **Define New Property** dialog box.

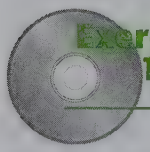


- **Default Offset** and **Comments.** Enter a value in the **Default Offset** edit box to specify the distance between two or more aligned balloons. The **Comments** edit box is used to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

PROFESSIONAL TIP



A single balloon style is all that is needed for most assembly drawings.



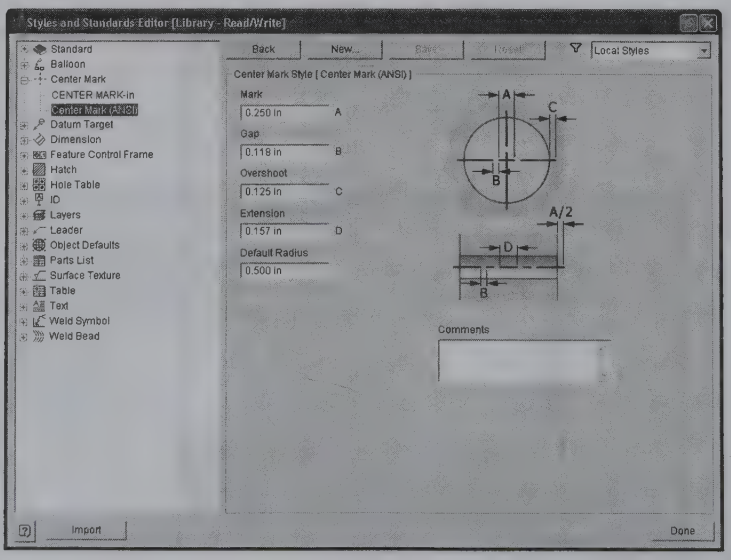
Exercise 16-6 Complete the Exercise on the Student CD.

Center Mark Styles

Drawings that contain circles, arcs, other round geometry, or symmetrical features require *center marks* or *centerlines* in order to be fully dimensioned and describe circular and symmetric geometry. Center mark styles are used to define center mark characteristics.

When you expand the **Center Mark** node in the **Styles and Standards Editor** browser, all the available center marks styles are listed as child nodes. Picking a specific center mark style displays variables used to set center marks, centerlines, centerline bisectors, and center patterns. See Figure 16-15. A centerline is made up of alternating short and long dashes, with a gap between the dashes. Short dashes typically cross at the center of a circle or arc. The center mark elements available in the **Center Mark Style** area correspond to the center mark options description area next to the edit boxes The top description displays the effects on a circle, while the bottom description displays the effects on a centerline. The **Center Mark Style** area includes the following variables:

Figure 16-15.
The **Center Mark Style** area of the **Styles and Standards Editor**.



- **Mark.** Defines the length of the short centerline dashes.
- **Gap.** Sets the gap, which is the space between the short and long centerline dashes.
- **Overshoot.** Sets the distance that centerlines extend past a circle or arc perimeter.
- **Extension.** Establishes the distance a symmetrical centerline extends past the feature, and controls the length relationship between short and long centerline dashes.
- **Default Radius.** This option identifies the center mark size for a suppressed feature in a pattern of features.
- **Comments.** Use this edit box to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

PROFESSIONAL TIP



A single center mark style is all that is required for most drawings.



Exercise 16-7 Complete the Exercise on the Student CD.

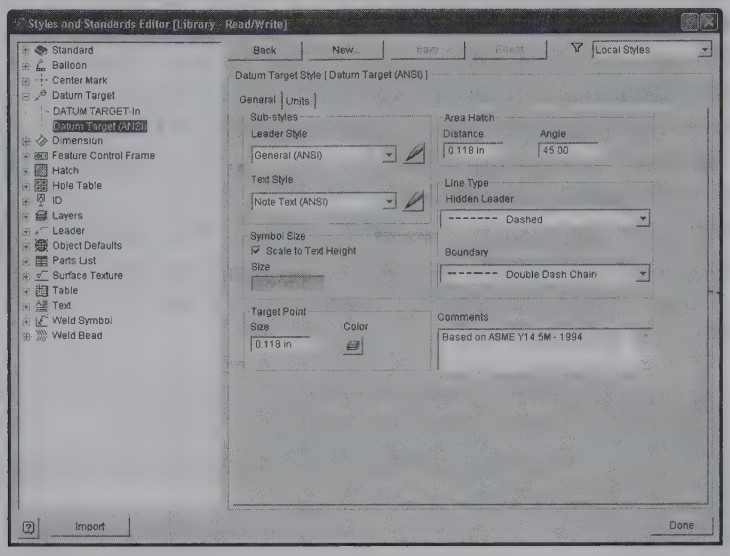
Datum Target Styles

Datum targets are used to specify points, lines, or areas of contact on a part that establishes datums when it is not possible or appropriate to use a surface. The datum target symbol is drawn as a circle with a horizontal line through the center. The top half of the circle is left blank unless the datum target symbol refers to a datum target area. In this case, the size of the datum target area is provided in the top half of the datum target symbol. The bottom half of the datum target symbol is used to identify the related datum with the datum reference letter and datum target number assigned sequentially to the datum, such as A1, A2, and A3. The datum target symbol is connected to the related datum target point, line, or area with a leader that does not have an arrowhead at the end. Datum target styles are used to define datum target characteristics.

Additional Datum Target Styles instruction and application are found on the CD accompanying this textbook. Figures 16-16 and 16-17 are placed with this discussion. When you expand the **Datum Target** node in the **Styles and Standards Editor** browser, all the available datum target styles are listed as child nodes. Picking a specific datum target style displays a **General** tab and a **Units** tab, as shown in Figure 16-16. The **General** tab contains the following general datum target settings:

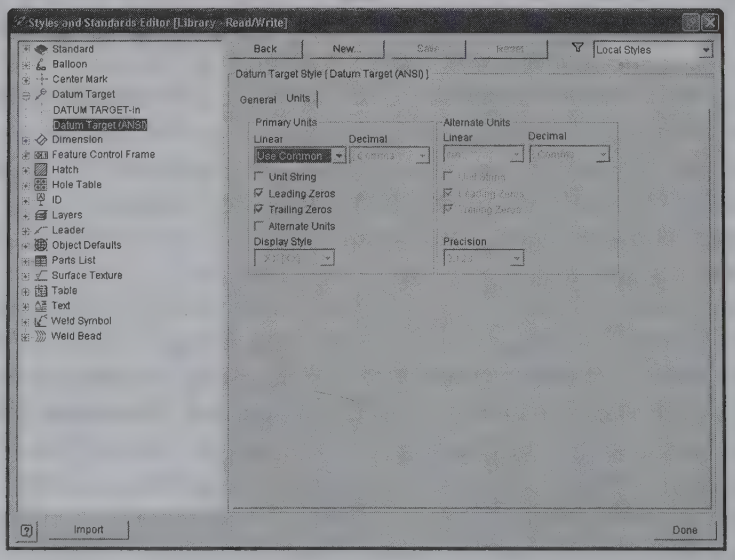
- **Sub-styles.** This area is used to select an existing leader and text style to use for datum targets and contains the following options:
  - **Leader Style.** Use this drop-down list to choose the leader style attached to datum target symbols. You can access the selected leader style variables by picking the **Edit Leader Style** button. Choosing the **Edit Leader Style** button displays the **Leader Style** area of the **Styles and Standards Editor**.
  - **Text Style.** Use this drop-down list to choose the text style used in the datum target symbol. You can access the selected text style variables by picking the **Edit Text Style** button. Choosing the **Edit Text Style** button displays the **Text Style** area of the **Styles and Standards Editor**.

Figure 16-16.
The **Datum Target Style** area of the **Styles and Standards Editor, General** tab.



- **Symbol Size.** This area is used to define the size of datum target symbol. You can choose to size the symbol according to the height of the text by selecting the **Scale to Text Height** check box. If you do not select the **Scale to Text Height** check box, the **Size** edit box becomes available allowing you to enter the exact size, or diameter, of the symbol.
- **Target Point.** This area allows you to specify the display options for point and line datum targets. Enter the desired point size in the **Size** edit box, and select a point color by picking the **Color** button. In most situations **Color** should be set to By Layer, which will automatically assign a text color based on the characteristics of the layer defined in the **Object Defaults Style** area for objects with text. However, you can choose to override the format by choosing a color other than By Layer.

Figure 16-17.
The **Datum Target Style** area of the **Styles and Standards Editor, Units** tab.



- **Area Hatch.** This area allows you to specify the hatch display options of circular and rectangular datum target areas. Enter the scale, or the distance between hatch lines, in the **Distance** edit box, and enter the hatch line angle in the **Angle** edit box.
- **Line Type.** This area contains a **Hidden Leader Line Type** drop-down list that allows you to define the type of leader line to use for far-side datum targets, and a **Boundary Line Type** drop-down list that specifies the type of line to use for the boundary, or outline, of circular and rectangular datum target areas.
- **Comments.** Use this edit box to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

The **Units** tab, shown in Figure 16-17, includes the following datum target unit settings:

- **Primary Units.** This area allows you to define the primary unit display characteristics of datum target symbol information, and includes the following options:
  - **Linear.** Use this drop-down list to specify the type of unit measurement, such as inch or mm. When you select **Use Common** from the **Linear** drop-down list, the units specified in the **Standard** area of the **Styles and Standards Editor**, described later in this chapter, are applied to datum target symbols. Selecting a unit of measurement other than **Use Common** overrides the default setting and provides access to the **Decimal** drop-down list.
  - **Decimal.** This drop-down list allows you to define the type of decimal marker, such as a period or comma, for primary datum target symbol units.
  - **Unit String.** Select this check box if you want the unit type to be displayed along with the datum target value, such as 12 mm.
  - **Leading Zeros.** Select this check box to display leading zeros, which changes the display of .5 to 0.5, for example.
  - **Trailing Zeros.** Select this check box to display trailing zeros, which changes the display of 45 to 45.0, for example.
  - **Alternate Units.** Pick this check box to display alternate units in datum target symbols. When you select the **Alternate Units** check box, the **Display Style** drop-down list and the **Alternate Units** area become active.
  - **Display Style.** Use this drop-down list to select and define the display characteristics of alternate units in reference to primary units.
- **Alternate Units.** As previously mentioned, this area becomes available when you pick the **Alternate Units** check box of the **Primary Units** area. The **Alternate Units** area allows you to further define the display characteristics of datum target symbol alternate units, and contains the following options:
  - **Linear.** Use this drop-down list to specify the type of alternate unit measurement, such as inch or mm. The purpose of alternate units is to show two types of dimension values, such as in and mm. Consequently, if you select in as the primary units in the **Primary Units** area, you should pick mm in the **Alternate Units** area, for example.
  - **Decimal.** This drop-down list allows you to define the type of decimal marker, such as a period or comma, for alternate datum target symbol units.
  - **Unit String.** Select this check box if you want the unit type to be displayed along with the datum target symbol value, such as 12 mm.
  - **Leading Zeros.** Select this check box to display leading zeros, which changes the display of .5 to 0.5, for example.

- **Trailing Zeros.** Select this check box to display trailing zeros, which changes the display of 45 to 45.0, for example.
- **Precision.** Use this drop-down list to specify the alternate unit precision. Precision identifies the accuracy of alternate unit values, by placing more or fewer numbers after the decimal point when using a decimal format, or a higher or lower denominator when using fractional format. For example, .125" has a higher precision than .13", and 1/16" has a higher precision than 1/8".

NOTE



Many of the datum target properties defined in the **Datum Target Style** area of the **Styles and Standards Editor** dialog box cannot be overridden when you place datum targets in a drawing.

PROFESSIONAL TIP



Due to the ability to control the display of all datum target options using a single datum target style, one datum target style is all that is required for most drawings.



Exercise 16-8 Complete the Exercise on the Student CD.

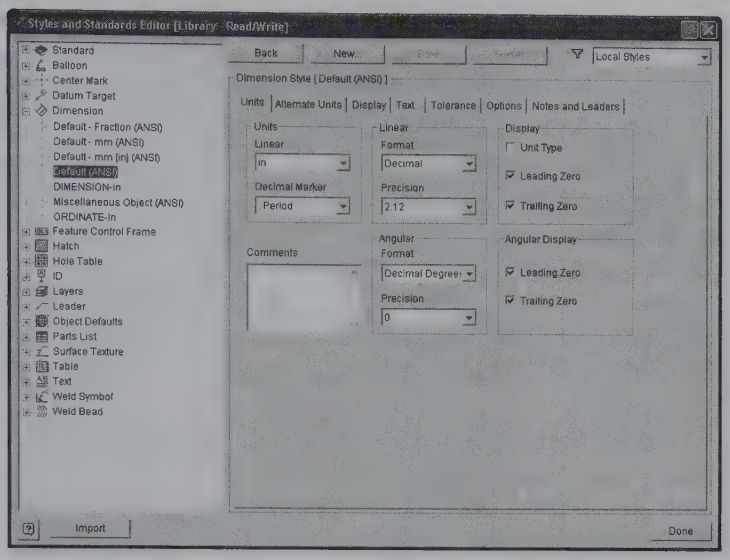
Dimension Styles

Dimension styles allow you to control most dimension display characteristics, including units, text, line display, and other dimension items. Dimension styles are used to define dimension characteristics.

When you expand the **Dimension** node in the **Styles and Standards Editor** browser, all the available dimension styles are listed as child nodes. Picking a specific dimension style displays dimension style variables, and the following tabs, as shown in Figure 16-18. The **Units** tab controls a variety of dimension unit specifications. The following options are available inside the **Units** tab:

- **Units.** Use this area to set the units for the dimension style. Specify the type of unit measurement, such as in or mm, from the **Linear** drop-down list. You can also define the type of decimal marker, such as a period or comma, from the **Decimal Marker** drop-down list.
- **Linear.** This area allows you to set the desired unit format and precision for linear dimensions. You can choose either a Decimal or Fractional unit format from the **Format** drop-down list. Then, specify the linear unit precision by selecting a value from the **Precision** drop-down list. **Precision** identifies the accuracy of the dimension by placing more or fewer numbers after the decimal point when using a decimal format, or a higher or lower denominator when using fractional format. For example, .125" has a higher precision than .13", and 1/16" has a higher precision than 1/8".
- **Display.** Use this area to display or suppress the unit type, leading zeros, and trailing zeros for linear dimensions. If you want the unit type to be displayed along with the dimension value, such as 12 mm, pick the **Unit Type** check box. Select the **Leading Zero** check box to display the leading zero, which changes

Figure 16-18.
The **Dimension Style** area of the **Styles and Standards Editor, Units** tab.



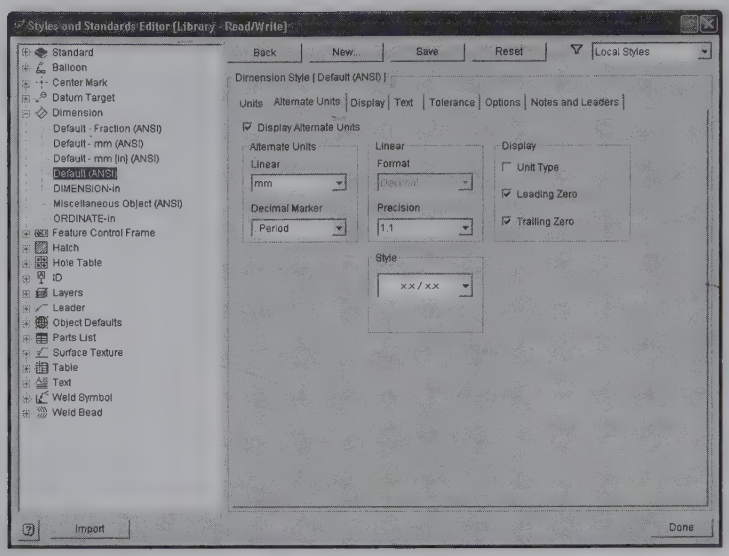
the display of .5 to 0.5, for example. Select the **Trailing Zero** check box to display the trailing zero, which changes the display of 45 to 45.0, for example.

- **Angular.** This area allows you to set the desired unit format and precision for angular, or nonlinear, dimensions. You can choose either a **Decimal Degrees** or **Deg-Min-Sec** format from the **Format** drop-down list. Then, specify the angular unit precision by selecting a value from the **Precision** drop-down list. Again, precision identifies the accuracy of the dimension by placing more or fewer numbers after the decimal point when using a **Decimal Degrees** format, or degrees, minutes, and seconds when using a **Deg-Min-Sec** format. For example, 45.5° has a higher precision than 45°, and 36°29′ 10″ has a higher precision than 36°29′ or 36°.
- **Angular Display.** Use this area to control the display, or suppression, of leading and trailing zeros for angular dimensions. Select the **Leading Zero** check box to display the leading zero, and pick the **Trailing Zero** check box to display the trailing zero.
- **Comments.** Use this edit box to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

The **Alternate Units** tab, shown in Figure 16-19, includes options that allow you to display alternate dimension units in addition to the units specified in the **Units** tab. If you want to display alternate units in your dimensions, pick the **Display Alternate Units** check box, making the following areas available:

- **Alternate Units.** Specify the type of alternate unit measurement, such as in or mm, from the **Linear** drop-down list, and the type of decimal marker, such as a period or comma, from the **Decimal Marker** drop-down list. The purpose of alternate units is to show two types of dimension values, such as in and mm. Consequently, if you select in as the primary units in the **Units** tab, you should pick mm in the **Alternate Units** tab, for example.
- **Linear.** This area allows you to set the desired alternate unit format and precision for linear dimensions. You can choose either a Decimal or Fractional unit format from the **Format** drop-down list, depending on the application. Then, specify the linear unit precision by selecting a value from the **Precision** drop-down list.

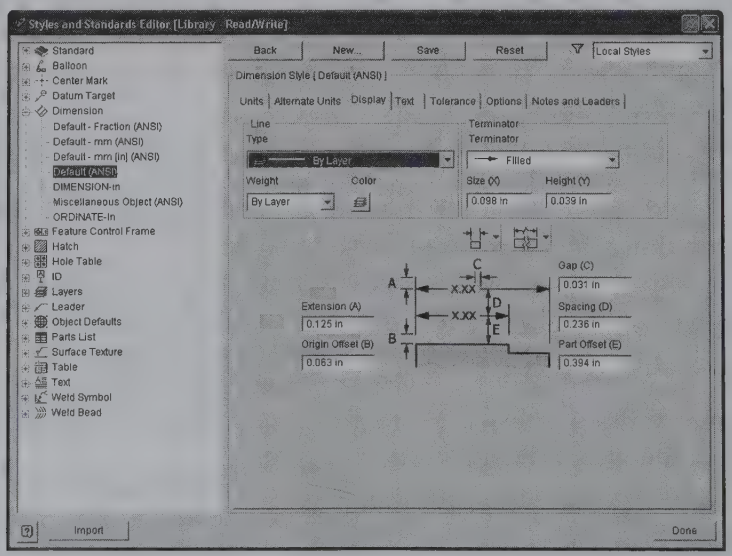
Figure 16-19.
The **Dimension Style** area of the **Styles and Standards Editor, Alternate Units** tab.



- **Display.** Use this area to display or suppress the unit type, leading zeros, and trailing zeros for alternate linear dimensions. If you want the alternate unit type to be displayed along with the dimension value, such as 12 mm, pick the **Display Unit Type** check box. Select the **Leading Zero** check box to display the leading zero, and the **Trailing Zero** check box to display the trailing zero.
- **Style.** This area is used to define the display characteristics of alternate units in reference to the primary units. Select the type of unit style format you want to use from the **Format** drop-down list.

The **Display** tab, shown in Figure 16-20, contains the following options that allow you to control a number of dimension display characteristics:

Figure 16-20.
The **Dimension Style** area of the **Styles and Standards Editor, Display** tab.



- **Line.** Use this area to define the appearance of dimensioning lines. In most situations the line **Type**, **Weight**, and **Color** variables should be set to By Layer. This will automatically assign a line type, line weight and color to the dimension style based on the characteristics of the layer defined in the **Object Defaults Style** area for objects with dimensions. However, you can choose to override the line variables by choosing a line type, line weight, and color other than By Layer.
- **Terminator.** This area allows you to define the style, size, and aspect of the dimension line terminator, such as an arrowhead. To define the terminator style, select the desired arrowhead from the **Terminator** drop-down list. Then, enter the size and aspect of the terminator in the **Size (X)** and **Height (Y)** edit boxes. ***Size*** defines the length of the arrowhead from the point to the end, while ***aspect*** specifies the width or thickness of the arrowhead in reference to the size. For example, most arrowheads are drawn at a 3:1 ratio. As a result, the size of the arrowhead would be 3 mm, or .125", while the aspect would be 1 mm, or .042".
- **Dimension Line Visibility.** This flyout button controls whether or not a dimension line is displayed when the dimension arrowheads are placed outside of the extension lines. Pick the **No Dimension Line** option if you do not want to place a dimension line between extension lines when the arrowheads are outside of the extension lines, or select the **Show Dimension Line** option if you do want to place a dimension line between extension lines when the arrowheads are outside of the extension lines.
- **Break Symbol.** Use this flyout button to add a break symbol to dimension lines when you dimension broken view geometry. Select the **Use Symbol** option to show the break symbol, or the **No Symbol** option to not add the symbol.
- **Extension.** Enter a value in this edit box to specify the length of the extension line extension from the dimension line.
- **Origin Offset.** Specify a value in the edit box to define the extension line offset, or gap, between the extension line and the object.
- **Gap.** Use this edit box to specify the distance, or gap, between the text and the dimension line.
- **Spacing.** Enter a value in this edit box to specify the distance, or spacing, between dimension lines.
- **Part Offset.** Enter a value in this edit box to define the distance, or spacing, between the object and the first dimension line parallel to the object.

NOTE

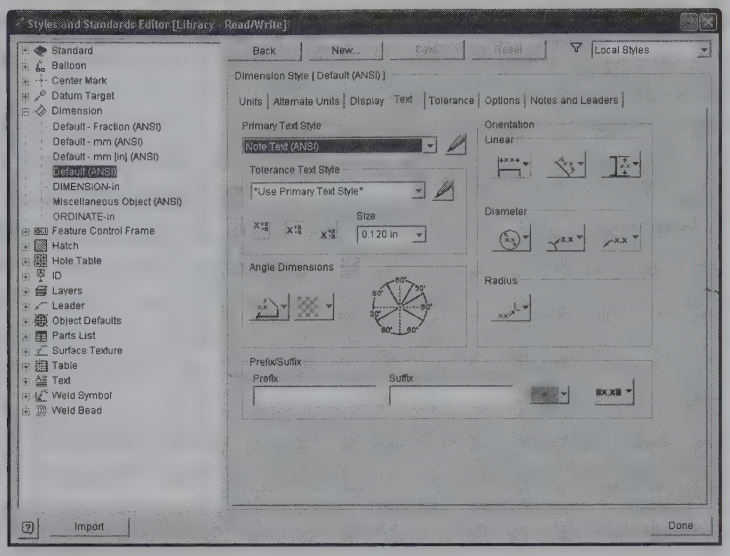


Dimension line spacing does not automatically occur. When you place or move a dimension, it becomes highlighted at the spacing locations specified in the **Display** tab. You must select the highlighted location.

The **Text** tab, shown in Figure 16-21, contains options that allow you to manage dimension value display preferences. Use the following options to define various types of dimension text appearance characteristics:

- **Primary Text Style.** Use this drop-down list to choose the text style used for primary dimension values. You can access the selected text style variables by picking the **Edit Text Style** button. Choosing the **Edit Text Style** button displays the **Text Style** area of the **Styles and Standards Editor**.
- **Tolerance Text Style.** This area includes options for defining the format of tolerance values. Use the drop-down list to choose the text style used for

Figure 16-21.
The **Dimension Style** area of the **Styles and Standards Editor, Text** tab.



tolerance dimension values. For most applications the *Use Primary Text Style* option is effective, though you can choose any available text style and access the selected text style variables by picking the **Edit Text Style** button. Choosing the **Edit Text Style** button displays the **Text Style** area of the **Styles and Standards Editor**. You also have the option to override the height of the tolerance value text by selecting a height from the **Size** drop-down list. The text sizes listed are specified in the **Standard** area of **Styles and Standards Editor**, described later in this chapter. You can select any available text size depending on the desired text height, or you can create a custom text height using the **Standard** area of the **Styles and Standards Editor**, then return to the **Dimension Style Text** tab to select the new value. Finally, the **Tolerance Text Style** area also contains top, middle, and bottom justification buttons, which allow you to further specify the tolerance text placement.

- **Angle Dimensions.** This area provides two flyout buttons that allow you to further specify the placement of text for specific applications when using angular dimensions:
  - **Angle Dimension Orientation.** Pick the **Inline - Horizontal** button to place unidirectional angular dimension text that is inside the dimension line. Select the **Inline - Aligned** button to place angular dimension text that is inline with dimension lines. Pick the **Parallel – Aligned** button to place angular dimension text that is aligned with and above the dimension line. You can also choose the **Parallel – Horizontal** button to place unidirectional angular dimension text that is outside the dimension line.
  - **Angle Dimension Modifier.** Choose the **Above for Horizontal Dimension** radio button to place angular dimension text above the extension line, when text is unidirectional. Pick the **Outside Dimension Line** radio button to place angular dimension text outside the dimension line. Select the **Outside for Angles 30° to 210°** radio button to place angular dimension text outside the dimension line when the object being dimensioned is between 30° and 210°.
- **Prefix/Suffix.** This area is used to add a prefix and a suffix to dimension text. A ***prefix*** is a word, character, or symbol added before or on top of the dimension value, while a ***suffix*** is a word, character, or symbol located after or below the

dimension value. Prefixes and suffixes are automatically added to dimension text when a dimension is created. If you want to add a prefix or a suffix to dimension values, enter the prefix in the **Prefix** edit box, and the suffix in the **Suffix** edit box. You may also choose to add symbols to the **Prefix** and **Suffix** edit boxes by selecting the desired symbol from the **Symbol** flyout button.

NOTE

If you do not want to add a prefix or a suffix to the dimension text, leave the edit boxes clear.

- **Orientation.** This area contains several flyout buttons that allow you to control the orientation and location of text in reference to various types of geometry and dimension information. These flyouts are grouped under the titles **Linear**, **Diameter**, and **Radius**. Graphics on the buttons help identify the desired orientation, while holding the cursor over the flyout button gives the button name. See Figure 16-22.
- **Tolerance.** This tab, shown in Figure 16-23, allows you to specify tolerancing options for drawing applications that require dimension tolerances. A *tolerance* is the total variation permissible in a size or location dimension. To define a dimension style with tolerances, first select the tolerance type or method from the **Method** drop-down list. Options in the **Tolerance** tab become active depending on the method you choose. If you select the **Default** tolerance

Figure 16-22.
These flyouts are from the **Orientation** area of the **Styles and Standards Editor**, **Text** tab.

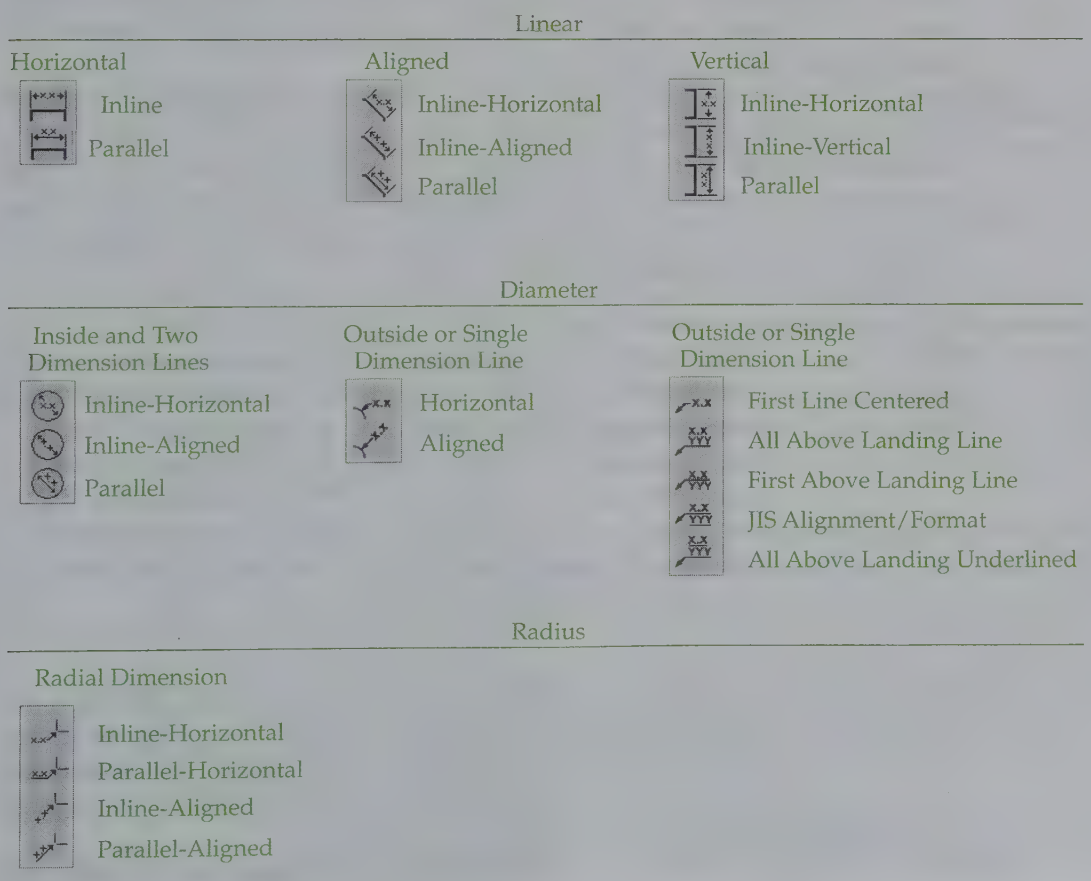
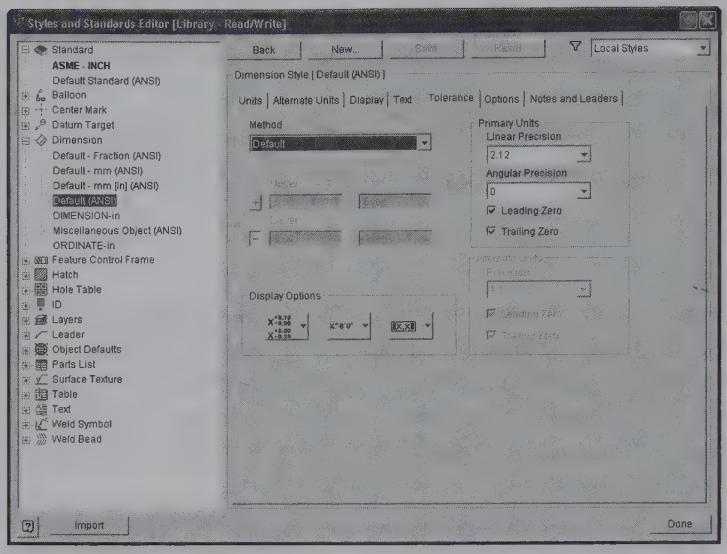


Figure 16-23.
The **Dimension Style** area of the **Styles and Standards Editor, Tolerance** tab.



method, no tolerance is applied to the dimension style. Once you select the desired tolerance method, use the following options to fully define the tolerance specifications:

- **Upper.** This edit box is available when you define limit tolerance methods such as Symmetrical, Deviation, Limits - Stacked, and Limits - Linear. Enter a value in the **Upper** edit box to set an upper value for a tolerance with a value range. For example, an object that is 8" long and has an upper range of .05" has an upper tolerance limit of 8.05".
- **Lower.** This edit box is available when you define limit tolerance methods such as Symmetrical, Deviation, Limits - Stacked, and Limits - Linear. Enter a value in the **Lower** edit box to set a lower value for a tolerance with a value range. Using the previous example, an object with a lower range of .05" has a lower tolerance limit of 7.95". The tolerance for this dimension reads 8.00 ±.05 for a Symmetric tolerance method option.

NOTE



When using the Deviation tolerance method option, both the **Upper** and **Lower** edit boxes are available. The normal **Deviation** application is used to set up an equal bilateral tolerance such as .6256±.005 or an unequal bilateral tolerance such as .6256+.005/-.002. To accomplish this, you need to select the minus (-) symbol in front of the value in the **Lower** edit box, the plus (+) symbol in front of the value in the **Upper** edit box.

- **Hole.** This drop-down list is available when using limits and fits tolerance methods for dimensioning holes. The hole values, A1 or B2 for example, available in the **Hole** drop-down list correspond to specific hole tolerances established in ANSI size and fit tables. For example, a hole dimensioned 20 H9 takes the place of 20+0.05/0.
- **Shaft.** This drop-down list is available when using limits and fits tolerance

methods for dimensioning shafts. The shaft values, al or b2 for example, available in the **Shaft** drop-down list correspond to specific hole tolerances established in ANSI size and fit tables. For example, a shaft dimensioned 10 d9 takes the place of 9.95 0/–0.05.

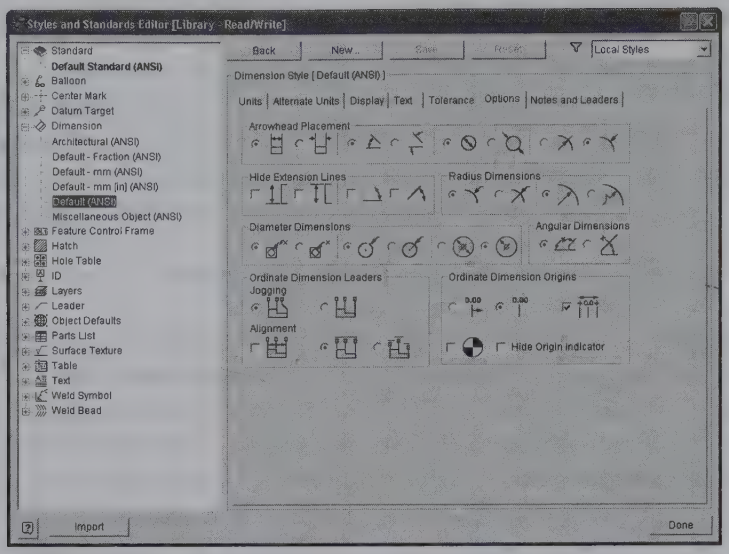
NOTE



When you select a value from the **Hole** drop-down list, the **Shaft** value is N/A. Similarly, when you select a value from the **Shaft** drop-down list, the **Hole** value is N/A, because the ANSI size and fit tolerances apply to either the hole or shaft.

- **Primary Units.** This area controls a variety of tolerance unit specifications. The following options are available inside the **Primary Units** area.
  - **Linear Precision.** Use this drop-down list to specify the linear tolerance unit precision. Precision identifies the accuracy of the tolerance by placing more or fewer numbers after the decimal point when using a decimal format, or a higher or lower denominator when using fractional format. For example, .125" has a higher precision than .13", and 1/16" has a higher precision than 1/8".
  - **Angular Precision.** Use this drop-down list to specify the angular tolerance unit precision. Again, precision identifies the accuracy of the tolerance by placing more or fewer numbers after the decimal point when using a Decimal degree format, or degrees, minutes, and seconds when using a Deg-Min-Sec format. For example, 45.5° has a higher precision than 45°, and 36°29′ 10″ has a higher precision than 36°29′ or 36°.
  - **Leading Zero.** Select this check box to display the leading zero, which changes the display of .5 to 0.5, for example.
  - **Trailing Zero.** Pick this check box to display the trailing zero, which changes the display of 45 to 45.0, for example.
- **Alternate Units.** This area controls the precision, and display of leading and trailing zeros of tolerances, when using alternate dimension units. The **Precision** drop-down list, **Leading Zero** check box, and **Trailing Zero** check box function like those available in the **Primary Units** area.
- **Display Options.** This area includes the following three flyout buttons that allow you to further control the display of dimension tolerances:
  - **Zero Tolerance Display.** This flyout button controls the display of unilateral dimension tolerances. Pick the **Full Display** option to show the plus and minus symbols, a leading zero if used, and trailing zeros if used. Select the **No Trailing Zeros** option to show the plus and minus symbols and a leading zero if used. Pick the **No Trailing Zeros – No Sign** option to show only a leading zero if used. Choose the **Suppress Display** option to hide all signs and zeros.
  - **Zero Arc Min/Sec Display.** Pick the **Show Arc Min/Sec** option to display minutes and seconds when using a Deg-Min-Sec format, or choose the **Hide Arc Min/Sec** option to display minutes and seconds when using a Deg-Min-Sec format.
  - **Basic Dimension Display.** Select the **Prefix/Suffix Inside** option to add a basic dimension rectangle around the dimension value and the prefix and suffix if used, or pick the **Prefix/Suffix Outside** option to add a basic dimension rectangle around just the dimension value.
- **Options.** This tab, shown in Figure 16-24, allows you to define additional dimension style options for dimensioning specific geometry. The **Options** tab

Figure 16-24.
The **Dimension Style** area of the **Styles and Standards Editor, Options** tab.

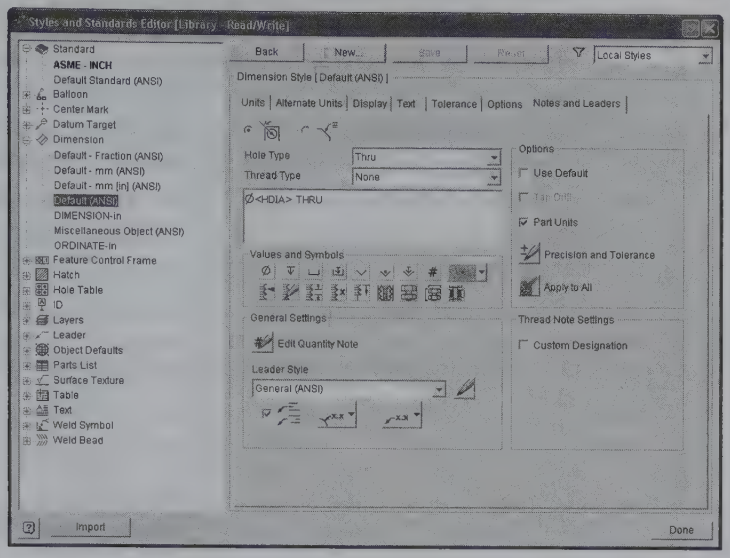


contains seven areas with radio buttons and check boxes. Specific dimensioning practices and applications are associated with each radio button or check box. A word, description, or tooltip is provided for each option, giving you an understanding of the option results. The following information briefly describes each area:

- **Arrowhead Placement.** This area contains two sections, each with two radio buttons. Pick the corresponding radio buttons to force dimension arrowheads inside or outside of linear and angular dimension extension lines.
- **Hide Extension Lines.** This area contains two sections, with four total check boxes. Select the appropriate check box or boxes to hide specific linear and angular dimension extension lines.
- **Radius Dimensions.** This area contains two sections, each with two radio buttons. Choose whether or not to force a leader from the center of a circular feature, and add a jog to the leader, when applying radial dimensions.
- **Diameter Dimensions.** This area contains three sections, each with two radio buttons. Choose the appropriate radio buttons based on the diameter dimensioning application. You can define whether or not a diameter symbol is added to the dimension, to force a leader from the center of a circular feature, and place the dimension value between two dimension lines.
- **Angular Dimensions.** Use this area to select the radio button corresponding to the desired angular dimension display, as shown in the illustration.
- **Ordinate Dimension Leaders.** This area contains options specific to ordinate dimensioning. Use the radio buttons and check box to define exactly how ordinate dimensions are aligned, and whether or not jogs and continuous descriptions are added.
- **Ordinate Dimension Origins.** Use this area to further define the format of ordinate dimension origins. Check boxes and radio buttons allow you to use an origin indicator, showing or hiding direction, and describe positive direction of dimensions.

- **Notes and Leaders.** This tab, shown in Figure 16-25, allows you to define the leader and annotation specifications that are applied when you place hole and

Figure 16-25.
The **Dimension Style** area of the **Styles and Standards Editor, Notes and Leaders** tab.



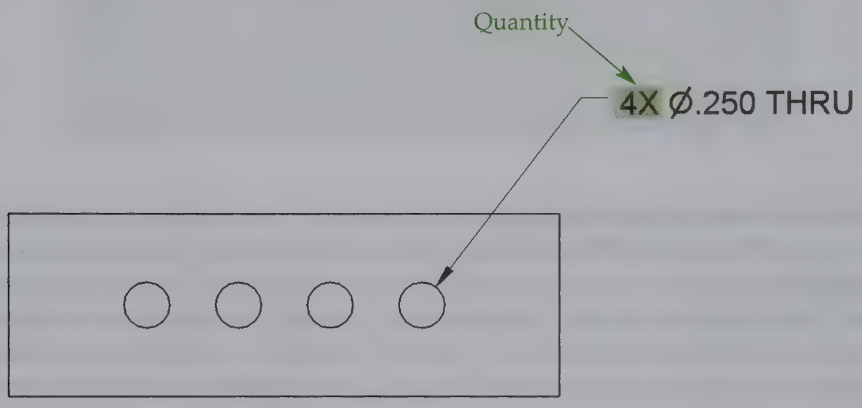
thread notes and chamfer notes. To define hole and thread note characteristics, pick the **Hole Note Settings** radio button. Next, select the hole type from the **Hole Type** drop-down list and the thread type from the **Thread Type** drop-down list. The combination of hole type and thread type determines the hole and thread note annotation, which you can modify. For example, if you want to adjust the annotation characteristics of notes placed on "thru" holes with full-length threads, pick the **Thru** hole type and the **Full Thread** thread type. Once you select the desired hole and thread type, use the following options to fully define hole and thread note settings:

- **Use Default.** Choosing this check box indicates that a predefined note is used for the selected hole and thread type. The default note is displayed in the **Hole/Thread Note** edit box. As long as the **Use Default** check box is selected, the default note cannot be modified, and you cannot place additional symbols or values. To define your own note characteristics for the selected hole and thread type, based on the default value, deselect the **Use Default** check box. When you deselect the **Use Default** check box, the **Hole/Thread Note** edit box becomes available for modification, and the buttons in the **Values and Symbols** area are selectable. The **Hole/Thread Note** edit box is much like any other edit box and allows you to create the desired hole and thread note. Use the buttons in the **Values and Symbols** area to add information to the note. Symbols selected from the **Values and Symbols** area represent the actual symbols used in the hole/thread note. For example, if you want to add a hole depth symbol, pick the **Hole Depth Symbol** button from the **Symbols** flyout button. Then, when you place the hole/thread note on a feature in the drawing, the proper symbol is displayed at the specified location in the note. Values selected from the **Values and Symbols** area represent the value of the hole or thread characteristic. For example, if you want to add a hole diameter value, pick the **Hole Diameter Value** button from the **Values and Symbols** area. Though the hole diameter is defined as <HDIA> in the **Hole/Thread Note** edit box, when you place the hole/thread note on a Ø5" feature, for example, 5" is displayed at the specified location in the note.

NOTE

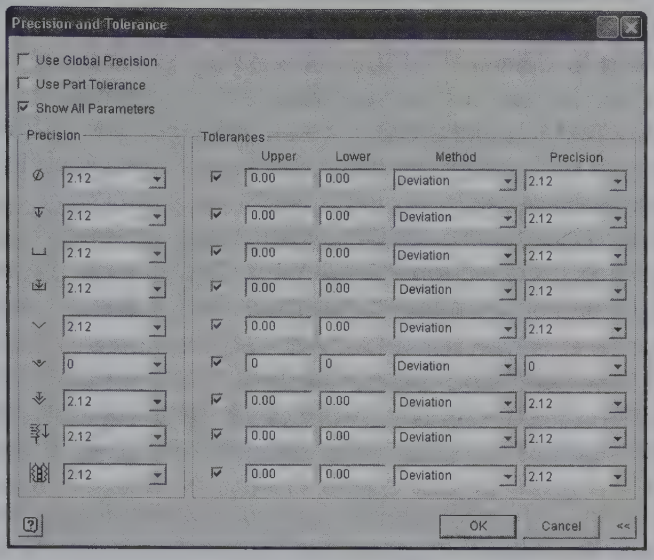
Use the **Quantity Note** symbol button, available from the **Values and Symbols** area, to add quantity information to multiple hole/thread features of a common size. See Figure 16-26. The **Quantity Note** symbol is fully parametric and is only applied to hole/thread notes in which two or more like features exist in the drawing view or pattern. The alternative to using the **Quantity Note** symbol is to manually enter a quantity by editing the default hole/thread note value, which is not a parametric operation.

Figure 16-26.
An example of using a numerator for multiple hole/thread features of a common size.



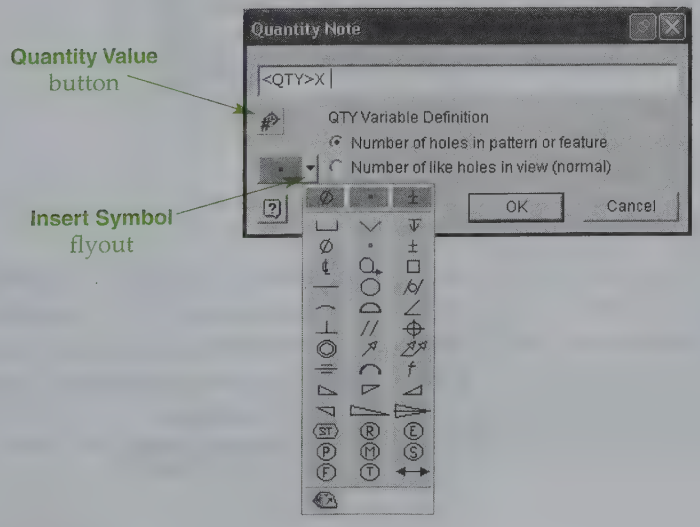
- **Tap Drill.** This check box is selectable depending on the type of thread you pick from the **Thread Type** drop-down list. Choose the **Tap Drill** check box if you want to include tap drill information from the thread data spreadsheet in the hole/thread note.
- **Part Units.** Pick this check box to use the measurement units specified in the **Document Settings** dialog box of the base view model file. For example, if you create a part model using inch units, the hole/thread note is displayed in inch units. Deselect the **Part Units** check box if you want to use the units defined in the **Dimension Styles** dialog box of the drawing file.
- **Precision and Tolerance.** Select this button to open the **Precision and Tolerance** dialog box, shown in Figure 16-27. The **Precision and Tolerance** dialog box allows you to define the precision and tolerance of selected hole types. Pick the **Use Global Precision** check box to use the same precision value defined in the **Units** tab of the **Dimension Styles** area. If you deselect the **Use Global Precision** check box, the drop-down lists in the **Precision** area become available for independent definition. Pick the **Use Part Tolerance** check box if you want to use the tolerances specified in the base view model file, or deselect the **Use Part Tolerance** check box, and pick check boxes in the **Tolerances** area. When you pick a check box in the **Tolerances** area, edit boxes and drop-down lists become available for setting tolerances for the selected hole type. Finally, when you pick the **Show All Parameters** check box, all hole parameters are shown even if they are not associated with the selected hole type.

Figure 16-27.
The **Precision and Tolerance** dialog box available when adjusting hole note settings.



- **Apply to All.** Choosing this button applies each of the preferences in the **Options** area to all hole and thread note types.
- **Custom Designation.** Pick this check box to use a custom thread designation created in the Thread.xls spreadsheet file, instead of using predefined, default thread note.
- **Edit Quantity Note.** Select this button to access the **Quantity Note** dialog box. See Figure 16-28. Use the **Quantity Note** dialog box to customize the value applied when you add a **Quantity Note** symbol, available from the **Values and Symbols** area, to the **Hole/Thread Note** edit box. You can choose to reference the number of holes according to a pattern of model features or the number of like holes in the drawing view by selecting the appropriate radio button. Additional symbols and quantity values can be added using the **Quantity Value** and **Insert Symbol** flyout buttons.

Figure 16-28.
The **Quantity Note** dialog box.

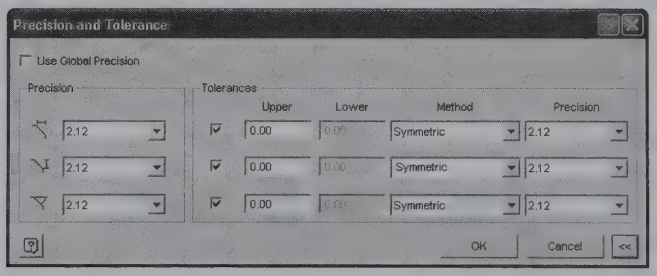


- **Leader Style.** Use this drop-down list to choose the leader style used for hole and thread notes. You can access the selected leader style variables by picking the **Edit Leader Style** button. Choosing the **Edit Leader Style** button displays the **Leader Style** area of the **Styles and Standards Editor**.
- **Justify Text to Leader.** Pick this check box to justify text according to the leader, which overrides the specified text style justification. For example, if the **Justify Text to Leader** check box is selected, text will left or right justify according to the location of the leader even if the text style justification is set to left, right, or center.
- **Leader Text Orientation.** This flyout button contains two options that allow you to place unidirectional leader text by picking the **Horizontal** button, or leader text that is aligned with the leader line by picking the **Aligned** button.
- **Leader Text Alignment.** This flyout button contains five options that define how leader text is aligned in reference to the leader shoulder. Pick the button that is appropriate for your application according to the button illustration.

To specify chamfer note characteristics, pick the **Chamfer Note Settings** radio button. Then, use the following options to fully define chamfer note settings:

- **Precision and Tolerance.** Select this button to open the **Precision and Tolerance** dialog box, shown in Figure 16-29. The **Precision and Tolerance** dialog box allows you to define the precision and tolerance of selected hole types. Pick the **Use Global Precision** check box to use the same precision value defined in the **Units** tab of the **Dimension Styles** area. If you deselect the **Use Global Precision** check box, the drop-down lists in the **Precision** area become available for independent definition.
- **Leader Style, Justify Text to Leader, Leader Text Orientation,** and **Leader Text Alignment** options function the same when adjusting chamfer note settings as when adjusting hole and thread note settings.

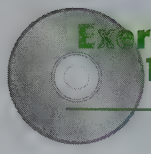
Figure 16-29.
The **Precision and Tolerance** dialog box available when adjusting chamfer note settings.



PROFESSIONAL TIP



Due to the ability to control a large number of dimension variables using a single dimension style, additional dimension styles may not be necessary. However, you can create as many dimension styles as required for unique dimensioning applications and standards.



Complete the Exercise on the Student CD.

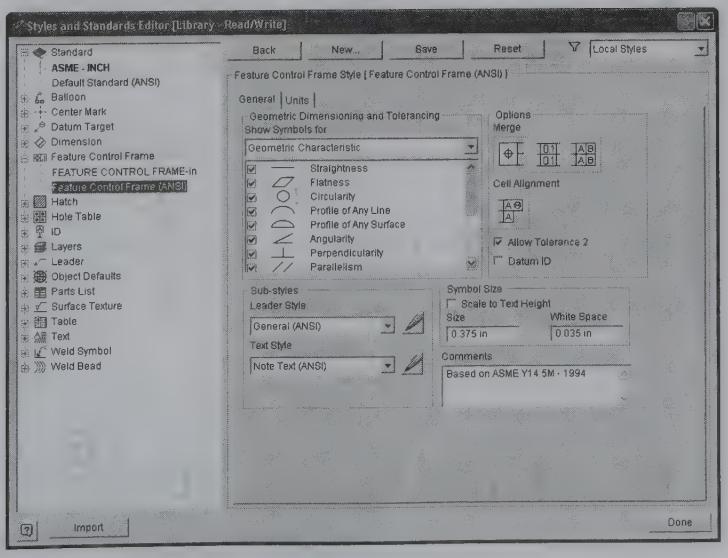
Feature Control Frame Styles

A ***feature control frame*** contains a geometric characteristic symbol, geometric tolerance, material condition symbol (when used), and datum reference (when used). The feature control frame is divided into compartments containing the geometric characteristic symbol in the first compartment, followed by the geometric tolerance in a compartment and each datum reference in a separate compartment. The geometric tolerance is preceded by a diameter symbol if the geometric tolerance zone is cylindrical. The feature control frame is generally placed with a related dimension on the drawing. Feature control frame styles are used to define feature control frames characteristics.

When you expand the **Feature Control Frame** node in the **Styles and Standards Editor** browser, all the available feature control frame styles are listed as child nodes. Picking a specific feature control frame style displays a **General** tab and a **Units** tab, as shown in Figure 16-30. Depending on the specified drafting standard, the options in the **Control Frame** tab are displayed slightly differently. However, the properties are available for all drafting standards and define the default feature control frame options when adding feature control frames. The **General** tab contains the following general feature control frame settings:

- **Geometric Dimensioning and Tolerancing.** This area contains a window with a number of check boxes that control the geometric dimensioning and tolerancing symbols available to use when placing feature control frames. If you do not want to have access to certain symbols in the **Feature Control Frame** dialog box, select the desired check boxes. Pick the **Geometric Characteristic** option from the **Show Symbols for** drop-down list to display geometric characteristic check boxes. Similarly, pick the **Material Removal Modifier** option from the **Show Symbols for** drop-down list to display material removal modifier check boxes.
- **Options.** This area contains the following settings:

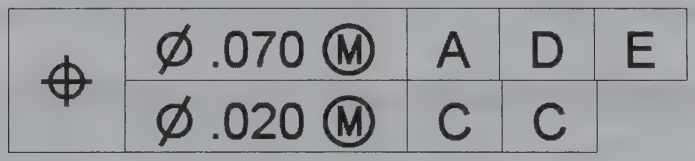
Figure 16-30.
The **Feature Control Frame Style** area of the **Styles and Standards Editor, General** tab.



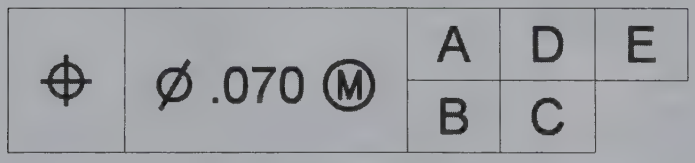
- **Merge.** This section contains three buttons that allow you to adjust the display of composite position and multiple single position tolerances. Pick the **Merge symbol** button to create a composite position tolerance. Select the **Merge tolerance** button to combine like tolerances. Pick the **Merge datum** button to combine like datums. See Figure 16-31.
- **Cell Alignment.** This section of the **Options** area contains the **Adjust Cells to Vertically Align** button, which as the name implies, vertically aligns composite position and multiple single position tolerances cells. See Figure 16-32.
- **Allow Tolerance 2.** Pick this check box in order to add an additional tolerance value to the **Feature Control Frame** dialog box.
- **Datum ID.** Pick this check box in order to add a new datum feature symbol to the **Feature Control Frame** dialog box.

- **Sub-styles.** This area is used to select existing leader and text styles to use for feature control frames and contains the following options:
  - **Leader Style.** Use this drop-down list to choose the leader style attached to feature control frame symbols. You can access the selected leader style variables by picking the **Edit Leader Style** button. Choosing the **Edit Leader Style** button displays the **Leader Style** area of the **Styles and Standards Editor**.
  - **Text Style.** Use this drop-down list to choose the text style used in feature control frames. You can access the selected text style variables by picking the **Edit Text Style** button. Choosing the **Edit Text Style** button displays the **Text Style** area of the **Styles and Standards Editor**.
- **Symbol Size.** This area is used to define the size of the feature control frame. You can choose to size the height of the symbol according to the height of the text by selecting the **Scale to Text Height** check box. If you do not select the **Scale to Text Height** check box, the **Size** edit box becomes available allowing you to enter the exact size, or height, of the symbol. A **White Space** edit box is also available which allows you to define the width of symbol, tolerance, and

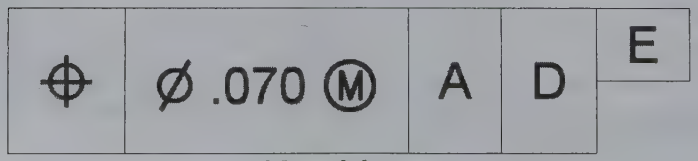
Figure 16-31.
The effects of merging like symbols, tolerances, and datums.



Merged symbols

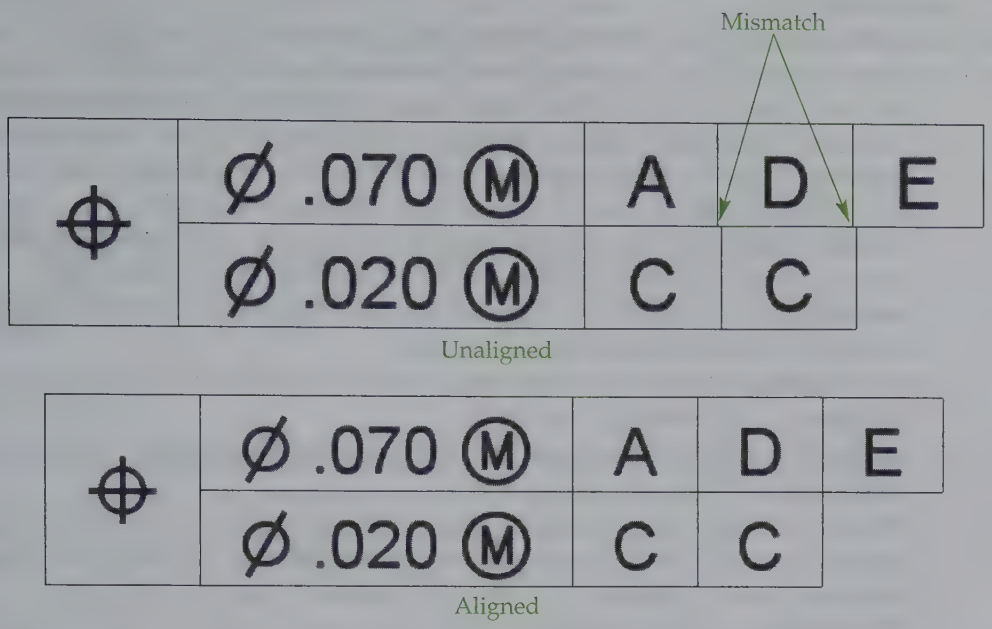


Merged tolerances



Merged datums

Figure 16-32.
Aligned cells of a feature frame have no mismatch.

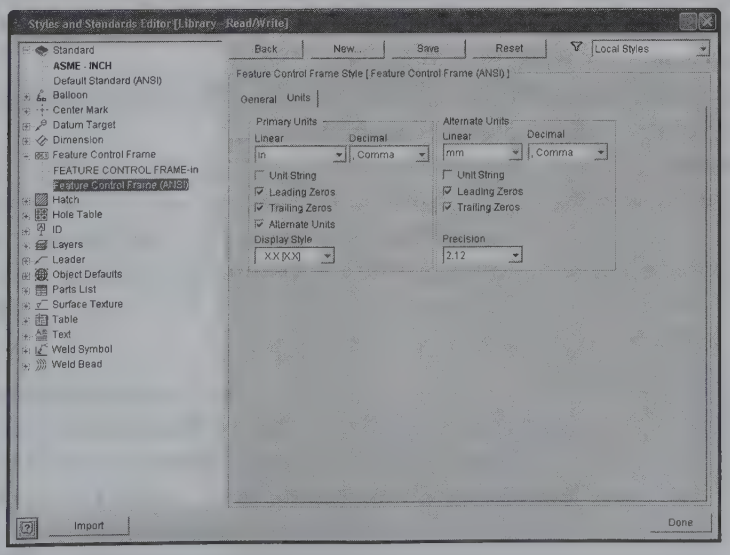


datum cells in reference to the amount of text in each cell. Increase or decrease the number in the edit box to create more or less white space, which widens or narrows the cells in reference to the text.

- **Comments.** Use this edit box to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

The **Units** tab, shown in Figure 16-33, includes the following feature control frame unit settings:

Figure 16-33.
The **Feature Control Frame Style** area of the **Styles and Standards Editor**, **Units** tab.



- **Primary Units.** This area allows you to define the primary unit display characteristics of feature control frame symbol information, and includes the following options:
  - **Linear.** Use this drop-down list to specify the type of unit measurement, such as inch or mm. When you select **Use Common** from the **Linear** drop-down list, the units specified in the **Standard** area of the **Styles and Standards Editor**, described later in this chapter, are applied to feature control frame symbols. Selecting a unit of measurement other than **Use Common** overrides the default setting and provides access to the **Decimal** drop-down list.
  - **Decimal.** This drop-down list allows you to define the type of decimal marker, such as a period or comma, for primary feature control frame units.
  - **Unit String.** Select this check box if you want the unit type to be displayed along with feature control frame values.
  - **Leading Zeros.** Select this check box to display leading zeros, which changes the display of .5 to 0.5, for example.
  - **Trailing Zeros.** Select this check box to display trailing zeros, which changes the display of 45 to 45.0, for example.
  - **Alternate Units.** Pick this check box to display alternate units in feature control frames. When you select the **Alternate Units** check box, the **Display Style** drop-down list and the **Alternate Units** area become active.
  - **Display Style.** Use this drop-down list to select and define the display characteristics of alternate units in reference to primary units.
- **Alternate Units.** As previously mentioned, this area becomes available when you pick the **Alternate Units** check box of the **Primary Units** area. The **Alternate Units** area allows you to further define the display characteristics of feature control frame symbol alternate units, and contains the following options:
  - **Linear.** Use this drop-down list to specify the type of alternate unit measurement, such as inch or mm. The purpose of alternate units is to show two types of dimension values, such as in and mm. Consequently, if you select in as the primary units in the **Primary Units** area, you should pick mm in the **Alternate Units** area, for example.
  - **Decimal.** This drop-down list allows you to define the type of decimal marker, such as a period or comma, for alternate feature control frame units.
  - **Unit String.** Select this check box if you want the unit type to be displayed along with feature control frame values.
  - **Leading Zeros.** Select this check box to display leading zeros, which changes the display of .5 to 0.5, for example.
  - **Trailing Zeros.** Select this check box to display trailing zeros, which changes the display of 45 to 45.0, for example.
  - **Precision.** Use this drop-down list to specify the alternate unit precision. Precision identifies the accuracy of alternate unit values, by placing more or fewer numbers after the decimal point when using a decimal format, or a higher or lower denominator when using fractional format. For example, .125″ has a higher precision than .13″, and 1/16″ has a higher precision than 1/8″.

NOTE



Many of the feature control frame properties defined in the **Feature Control Frame Style** area of the **Styles and Standards Editor** dialog box cannot be overridden when you place datum targets in a drawing.

PROFESSIONAL TIP

One feature control frame style is all that is required for most drawings because of the ability to control the display of feature control frame options using a single feature control frame style.

Exercise 16-10 Complete the Exercise on the Student CD.

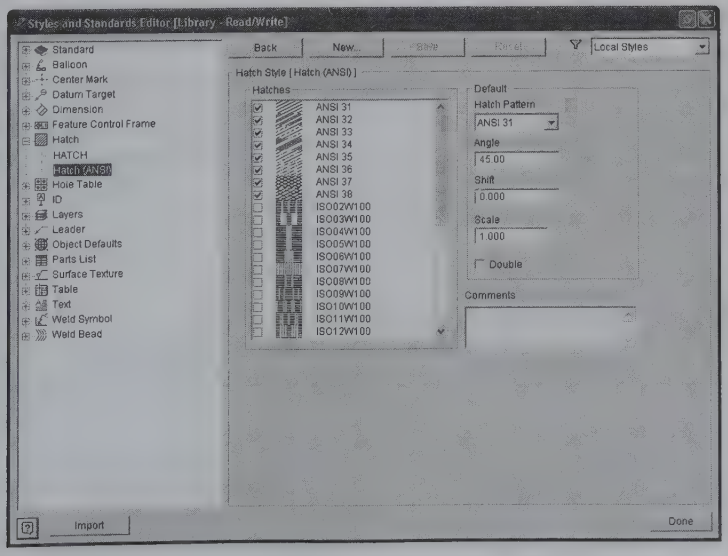
Hatch Styles

The term ***hatch*** or ***hatching*** is used by Inventor to describe a graphic pattern, including solid color fills. Hatching is primarily used when creating a sectional drawing view. However, there are a variety of applications such as applying a graphic pattern or solid color to border, title block, and symbol sketches. Hatch styles are used to control the default hatch pattern options when hatching a sketched region or a section view.

When you expand the **Hatch** node in the **Styles and Standards Editor** browser, all the available hatch styles are listed as child nodes. Picking a specific hatch style displays hatch style variables and the following areas, as shown in Figure 16-34:

- **Hatches.** This selection box contains all available hatch patterns. Hatch pattern check boxes that are selected are available when hatching a sketch region, or section, and can be modified. Select an additional hatch pattern for use by picking the check box next to the hatch name. Make a hatch pattern unusable by deselecting the check box.
- **Default.** This area allows you to define the default properties of the selected hatch pattern and contains the following options:
  - **Hatch Pattern.** Pick the hatch pattern you want to modify from this pull-down menu. Only hatch patterns selected in the **Hatches** area are available.

Figure 16-34.
The **Hatch Style** area of the **Styles and Standards Editor**.



- **Angle.** Use this edit box to define the angle of the hatch pattern. For example, a hatch pattern that consists of horizontal lines has an angle of 0° or 180°.
- **Shift.** Enter a value in this edit box to shift, or offset, the hatch pattern when hatching next to another hatched region. A shift does not align hatch patterns with each other.
- **Scale.** This edit box defines the dimensions of a hatch pattern. When you increase the scale, the hatch density decreases. When you decrease the scale, the hatch density increases. Enter the hatch pattern scale in the edit box, or pick the flyout button and select a scale.
- **Double.** Select this check box to create a double, or mirrored, hatch pattern. See Figure 16-35.

- **Comments.** Use this edit box to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

NOTE



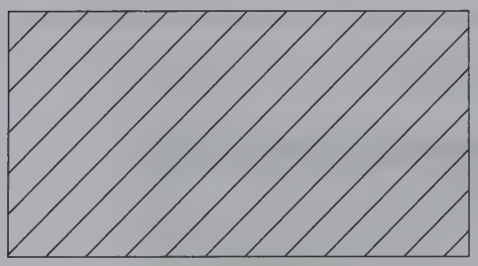
The hatch properties defined in the **Hatch Style** area of the **Styles and Standards Editor** are default and can be overridden when you place hatching.

PROFESSIONAL TIP

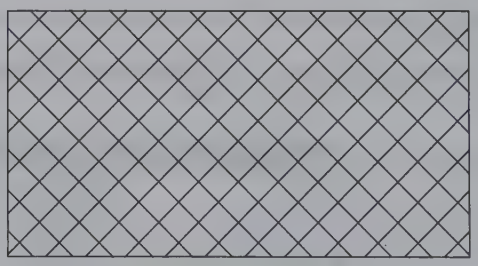


The purpose of the **Hatch Style** area of the **Styles and Standards Editor** is to adjust default hatch characteristics. Typically a single hatch style is all that is required, because most hatch characteristics are applied when you create a sectional view or add hatching.

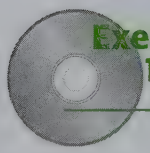
Figure 16-35.
Select the **Double** option to mirror the hatch pattern.



Single hatch pattern



Hatch pattern with the double option



Exercise 16-11 Complete the Exercise on the Student CD.

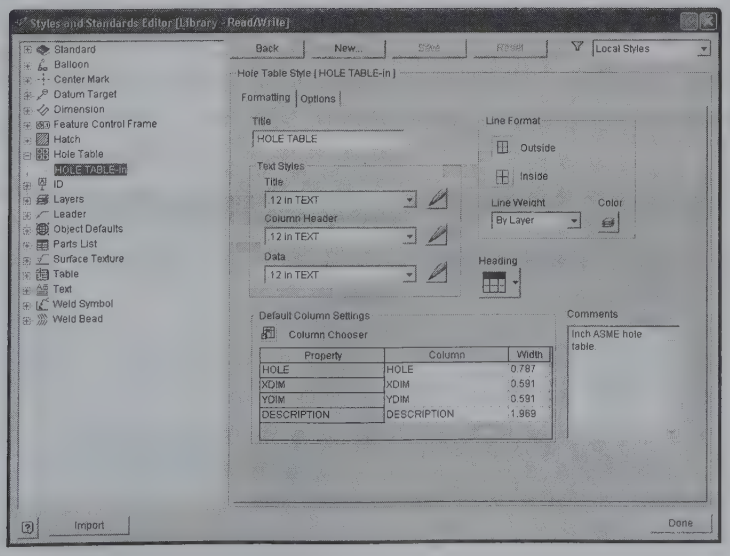
Hole Table Styles

Tabular dimensioning is a type of arrowless, ordinate dimensioning. Unlike standard ordinate dimensioning, tabular dimensioning involves a system in which coordinate dimensions and size dimensions are given in a table, correlating features on the drawing with a hole tag. Hole table styles are used to define tabular dimensioning table and tag display characteristics.

When you expand the **Hole Table** node in the **Styles and Standards Editor** browser, all the available hole table styles are listed as child nodes. Picking a specific hole table style displays a **Formatting** tab and an **Options** tab, as shown in Figure 16-36. The **Formatting** tab includes the following hole table formatting settings:

- **Title.** Use this edit box to redefine the hole table title.
- **Text Styles.** This area contains drop-down lists to format the table's title, column heads, and data.
  - **Title.** Use this drop-down list to choose the text style of the table's title. You can access the selected text style variables by picking the **Edit Text Style** button. Choosing the **Edit Text Style** button displays the **Text Style** area of the **Styles and Standards Editor**.
  - **Column Header.** Use this drop-down list to choose the text style of the table's column heads. You can access the selected text style variables by picking the **Edit Text Style** button. Choosing the **Edit Text Style** button displays the **Text Style** area of the **Styles and Standards Editor**.
  - **Data.** Use this drop-down list to choose the text style of the table's data. You can access the selected text style variables by picking the **Edit Text Style** button. Choosing the **Edit Text Style** button displays the **Text Style** area of the **Styles and Standards Editor**.
- **Line Format.** To change the line weight and color of the hole table border, pick the **Outside** button. Then select the desired line weight from the **Line Weight** drop-down list. Pick the **Color** button to access the **Color** dialog box, which allows you to change the hole table border color. To change the line weight and color of cell borders, which are the lines that separate table entries and titles, pick the **Inside** button. Then select the desired line weight from the **Line Weight** drop-down list and pick the **Color** button to access the **Color** dialog box, which again allows you to change the hole table border color.

Figure 16-36.
The **Hole Table Style** area of the **Styles and Standards Editor, Formatting** tab.



NOTE



In most situations, the Line Weight and Color variables should be set to By Layer. This will automatically assign a line weight and color to the hole table style based on the characteristics of the layer defined in the **Object Defaults Style** area for hole tables.

- **Heading.** This flyout button allows you to modify the location of the hole table column headings, or completely remove the headings if necessary. By default, the **Top** position button is selected, which means that the hole table heading is located at the top of the table. Pick the **Bottom** position button to move the heading to the bottom of the table, or pick the **No Heading** title position button to completely remove the hole table heading.
- **Default Column Settings.** This area contains the following options used to define default display characteristics of hole table columns:
  - **Column Chooser.** Pick this button to display the **Hole Table Column Chooser** dialog box, shown in Figure 16-37. The **Hole Table Column Chooser** dialog box has a purpose similar to the **Property Chooser** dialog box described earlier, and allows you to add columns to the hole table by referencing properties. Additional property types are shown in the **Available Properties** list box. To create a custom property, pick the **New Property** button, which opens the **Define New Property** dialog box. The next step to defining a new column is to specify which property or properties you want to add to the parts list. To accomplish this task, pick a property and select the **Add** button to add the property to the **Selected Properties** list box. When two or more properties are specified, the properties are displayed in the hole table in the same order shown in the **Selected Properties** list box. To move a property up in the list, highlight the property and pick the **Move Up** button. Conversely, to move a property down in the list, highlight the property and select the **Move Down** button. Once you finish specifying property information in the **Parts List Column Chooser** dialog box, pick the **OK** button to create the new column.
  - **Property.** This column shows all the properties used in the hole table. Right-clicking on any of the properties shows a pop-up menu containing the following options: **Column Chooser** and **Format Column**. Pick **Column Chooser** to open the **Hole Table Column Chooser** dialog box described earlier in this section. Pick **Format Column** to access the **Format Column** dialog box. See Figure 16-38.

Figure 16-37.
The **Hole Table Column Chooser** dialog box.

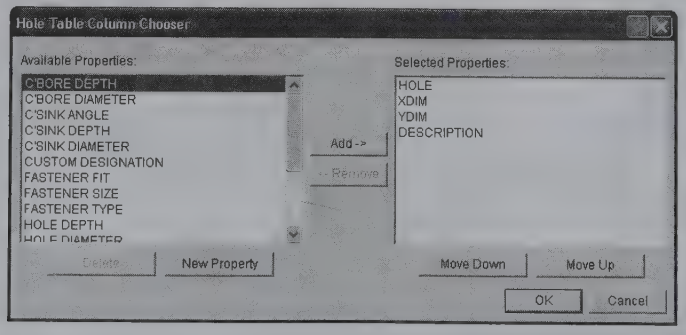
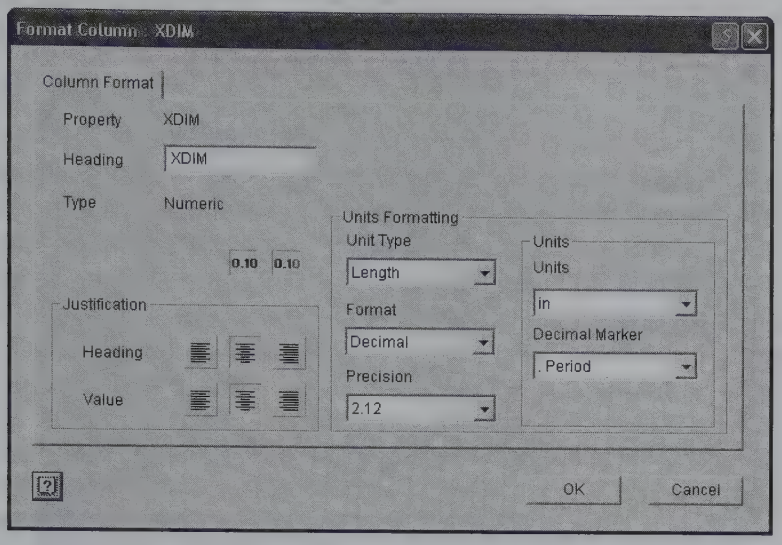


Figure 16-38.
The **Format Column** dialog box.



The **Format Column** dialog box contains variables that control the format of the property listed next to the **Property:** name. Use the **Heading** edit box to modify the column name. **Type** identifies the property column value category, such as text or numeric, as defined in the **Properties** dialog box. Select the **Leading Zeros** button to display leading zeros, which changes the display of .5 to 0.5, for example. Select the **Trailing Zeros** button to display trailing zeros, which changes the display of 45 to 45.0, for example.

The **Justification** area includes buttons that allow you to control the left, center, or right justification, or placement of heading and value text in reference to the column cell edges.

The **Units Formatting** area controls the units and display of numbers. Select **Length**, **Angularity**, **Volume**, or **Mass** from the **Unit Type** drop-down list before continuing with the other options in the **Units Formatting** area.

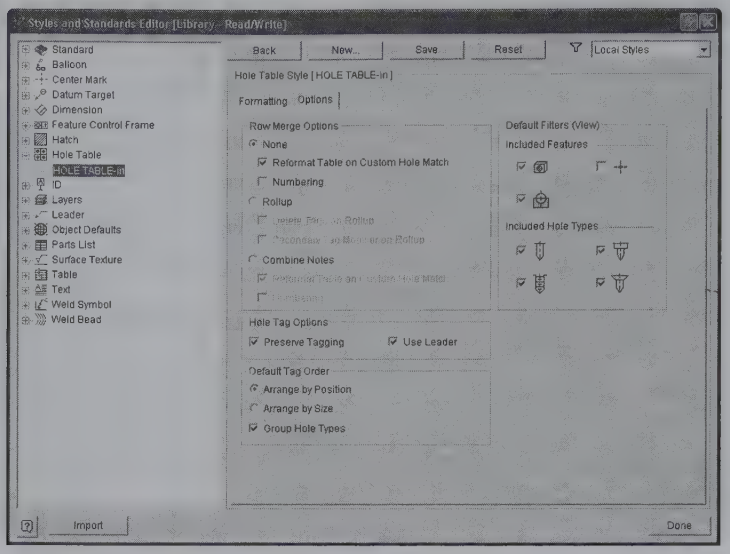
- **Column.** This column identifies the name used in the hole table for each property. To change the name, highlight the name you want to modify, enter a new name, and then select outside of the cell.
- **Width.** Use this column to define the width of each hole table property column. To change the width, highlight the value you want to modify, enter a new width, and then select outside of the cell.

- **Comments.** Use this edit box to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

The **Options** tab, shown in Figure 16-39, includes the following additional hole table options:

- **Row Merge Options.** This area allows you to control the display of separate holes, with the same properties, in the hole table. For example, if two holes are the same, you can choose to display a separate row for each hole, or merge the holes together into a single row. By default, the **None** radio button is selected, which means that holes are added to the hole table based on each hole's specific properties. Using the previous example, both holes would be displayed in the hole table on separate rows. You can choose to reformat the hole table when a custom hole match operation is applied by selecting the **Reformat Table on Custom Hole Match** check box, and replace alphabetical hole identification

Figure 16-39.
The **Hole Table Style** area of the **Styles and Standards Editor, Options** tab.



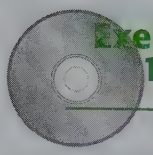
by picking the **Numbering** check box. To merge, or combine, like holes into a single row, pick the **Roll Up** radio button. Then, you can choose whether or not to delete hole tags by picking the **Delete Tags on Rollup** check box, and include a secondary numerator for hole tags by selecting the **Secondary Tag Modifier on Rollup** check box. Another option is to merge only description cells by selecting the **Combine Notes** radio button. The **Reformat Table on Custom Hole Match** and **Numbering** check box are available when you pick the **Combine Notes** radio button.

- **Default Filters (View).** This area contains several check boxes that allow the specified features and hole types to be included in the tabular dimension. As the name implies, deselecting the check boxes filters, or removes, the features and holes from the table.
- **Hole Tag Options.** Use this area to specify default hole tag settings. Pick the **Preserve Tagging** check box in order to keep the existing tagging format of tabular dimensions, when a new hole is added to the drawing. Select the **Leader** check box to attach a leader to hole tags that are dragged away from the hole.
- **Default Tag Order.** This area contains two radio buttons and a check box that allow you to set the default configuration of hole tag values. Pick the **Arrange by Position** radio button to define tag values according to the location of holes in the drawing view, or select the **Arrange by Size** radio button to define tag values in reference to hole size, from the smallest hole to the largest. You can also pick the **Group Hole Types** check box to define hole tag values based on hole type; then holes are further arranged according to the default tag order radio button you select.

PROFESSIONAL TIP



For most drawings, only one style for hole tables is all that is required, because of the ability to control the display of tabular dimensioning options.



Exercise 16-12 Complete the Exercise on the Student CD.

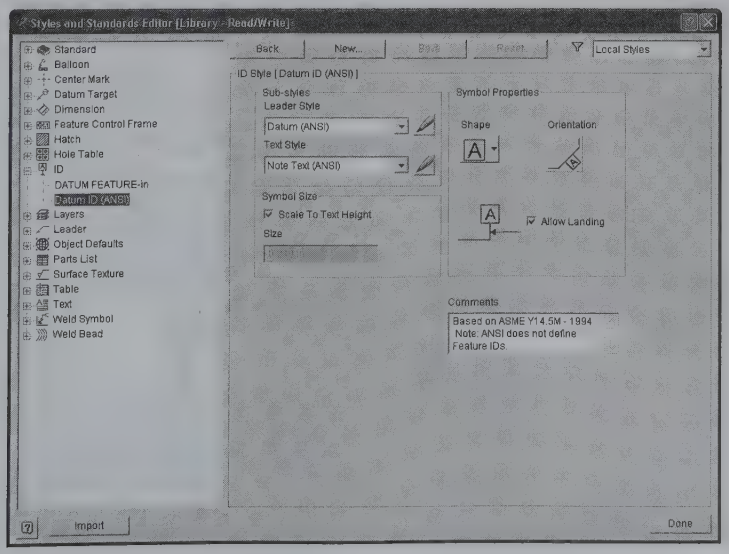
ID Styles

A ***datum feature symbol*** is also referred to as a ***datum identification symbol***, and by Inventor as ***ID***. ***Datums*** are considered theoretically perfect surfaces, planes, points, or axes, and are identified by datum feature symbols on the drawing. The datum feature symbol is a square box that contains the datum reference letter. The box is connected to a triangular datum arrow that contacts the datum surface or related application. Datum feature symbols are attached to the surface with a leader or off of an extension line, with a diameter or center plane dimension, or attached to a feature control frame. ID styles are used to define datum feature symbol characteristics.

When you expand the **ID** node in the **Styles and Standards Editor** browser, all the available ID styles are listed as child nodes. Picking a specific ID style displays ID style variables, and the following areas, as shown in Figure 16-40.

- **Sub-styles.** This area is used to select an existing leader and text style to use for datum feature symbols and contains the following options:
  - **Leader Style.** Use this drop-down list to choose the leader style attached to datum feature symbols. You can access the selected leader style variables by picking the **Edit Leader Style** button. Choosing the **Edit Leader Style** button displays the **Leader Style** area of the **Styles and Standards Editor**.
  - **Text Style.** Use this drop-down list to choose the text style used in the datum feature symbol. You can access the selected text style variables by picking the **Edit Text Style** button. Choosing the **Edit Text Style** button displays the **Text Style** area of the **Styles and Standards Editor**.
- **Symbol Size.** This area is used to define the size of datum feature symbols. You can choose to size the symbols according to the height of the text by selecting the **Scale to Text Height** check box. Or, if you do not select the **Scale to Text Height** check box, the **Size** edit box becomes available allowing you to enter the exact size, or height, of the symbol.

Figure 16-40.
The **ID Style** area of the **Styles and Standards Editor**.

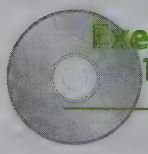


- **Symbol Properties.** This area provides additional datum feature symbol options. You can choose a shape other than a square by selecting a shape from the **Shape** drop-down list. Choose to display the feature control frame at any angle or in a unidirectional orientation by picking the **Orientation** button. You can also specify whether a leader landing, is available when placing datum feature symbols, by selecting or deselecting the **Allow Landing** check box.
- **Comments.** Use this edit box to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

PROFESSIONAL TIP



For most drawings one datum ID style is all that is required, because of the ability to control the display of all datum feature symbol options using a single ID style. Still, you can create additional ID styles if necessary.



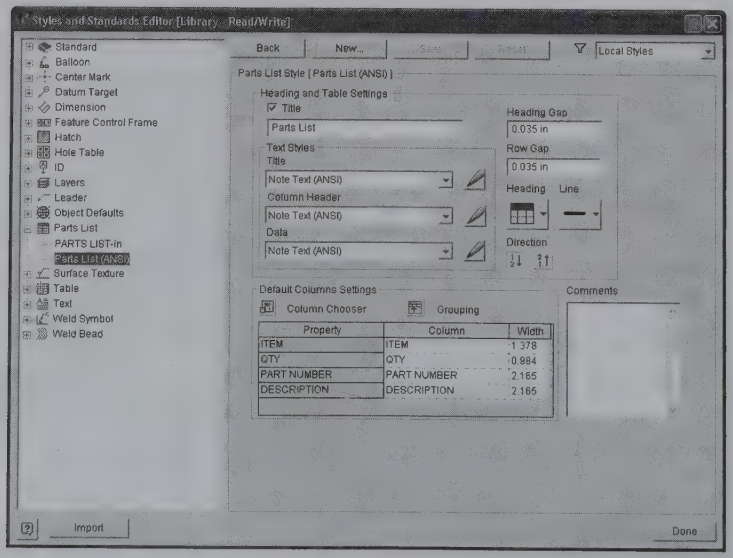
Exercise 16-13 Complete the Exercise on the Student CD.

Parts List Styles

A ***parts list*** records and displays the parts and subassemblies used to create an assembly, and keys or links balloons to component information. Parts list information differs greatly depending on the application and specific company standards. Parts list styles are used to define parts list characteristics.

When you expand the **Parts List** node in the **Styles and Standards Editor** browser, all available parts list styles are displayed as child nodes. Picking a specific parts list style displays parts list style variables, and the following areas, as shown in Figure 16-41.

Figure 16-41.
The **Parts List Style** area of the **Styles and Standards Editor**.



- **Heading and Table Settings.** This area contains the following parts list formatting options:
  - **Title.** Pick this check box to display the parts list title in the drawing and enter a title in the edit box.
  - **Text Styles**. This area contains drop-down lists to format the table's title, column heads, and data. Use the **Title, Column Header, and Data** drop-down lists to choose the text style of each of these table features. You can access the selected text style variables by picking the **Edit Text Style** button. Choosing the **Edit Text Style** button displays the **Text Style** area of the **Styles and Standards Editor**.
  - **Heading Gap.** Use this edit box to enter the amount of space between the parts list heading text and the heading cell.
  - **Row Gap.** Use this edit box to enter the amount of space between parts list text and cell edges.
  - **Heading.** This flyout button allows you to modify the location of the parts list heading, or completely remove the heading if necessary. By default, the **Top** position button is selected, which means that the parts list heading is located at the top of the table. Pick the **Bottom** position button to move the heading to the bottom of the table, or pick the **No Heading** position button to completely remove the parts list heading.
  - **Line Spacing.** This flyout button allows you to select and specify single, double, or triple line spacing between multiple lines of parts list text.
  - **Direction.** These buttons control the order of parts list items. Pick the **Add new parts to bottom** button to create a parts list that provides data from the bottom of the sheet to the top. Select the **Add new parts to top** button to create a parts list that provides data from the top of the sheet to the bottom. See **Figure 16-42.**

Figure 16-42.
Parts lists may start from the top or bottom.

PARTS LIST			
ITEM	QTY	PART NUMBER	DESCRIPTION
1	1	Body	
2	1	Pin	
3	1	Screw	
4	1	SWIVEL	C-CLAMP SWIVEL

4	1	SWIVEL	C-CLAMP SWIVEL
3	1	Screw	
2	1	Pin	
1	1	Body	
ITEM	QTY	PART NUMBER	DESCRIPTION
PARTS LIST			

- **Default Columns Settings.** This area contains the following options used to define default display characteristics of parts list column:
  - **Column Chooser.** Pick this button to display the **Parts List Column Chooser** dialog box, shown in Figure 16-43. The **Parts List Column Chooser** dialog box has a purpose similar to the **Property Chooser** and **Hole Table Column Chooser** dialog boxes described earlier, and allows you to add columns to the parts list by referencing properties. The first step in adding a property column is to select the property fields, from which you want to have access, using the **Select available properties from** drop-down list. The property types are shown in the **Available Properties** list box. If you pick the **All Properties** option, all property types are shown in the **Available Properties** list box. To create a custom property, pick the **New Property** button, which opens the **Define New Property** dialog box. The next step to defining a new column is to specify which property or properties you want to add to the parts list. To accomplish this task, pick a property and select the **Add** button to add the property to the **Selected Properties** list box. When two or more properties are specified, the properties are displayed in the parts list in the same order shown in the **Selected Properties** list box. To move a property up in the list, highlight the property and pick the **Move Up** button. Conversely, to move a property down in the list, highlight the property and select the **Move Down** button. Once you finish specifying property information in the **Parts List Column Chooser** dialog box, pick the **OK** button to create the new column.
  - **Grouping.** Pick this button to access the **Group Settings** dialog box, shown in Figure 16-44. The **Group Settings** dialog box allows you to control the display of separate assembly drawing components with the same properties in the parts list. For example, if two brackets used in an assembly have the same part number but come from different vendors, you can choose to display a separate row for each part, or merge the parts together into a single row. By default, the **Group** check box is not selected, which means that assembly drawing components are added to the parts list based on each component's specific properties. Using the previous example, both brackets are displayed in the parts list on separate rows. To group, or combine, similar components, such as the brackets, into a single row, pick the **Group** check box. Once the **Group** check box is selected, you can group some assembly drawing components, and separate others, by selecting the first property that differentiates the items from the **First Key** drop-down list. Then if needed, select a second and even third property to differentiate the items, using the **Second Key** and the **Third Key** drop-down lists. You can also

Figure 16-43.
The **Parts List Column Chooser** dialog box.

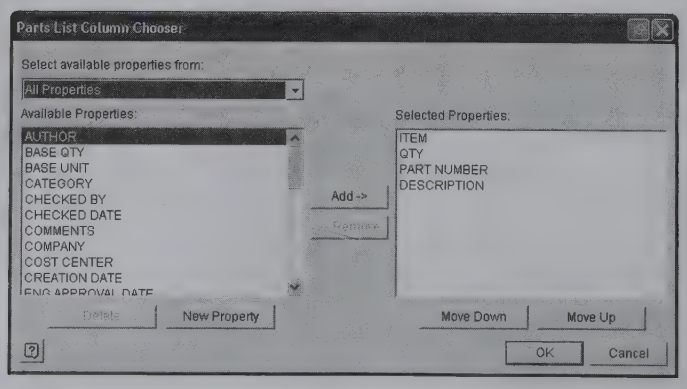
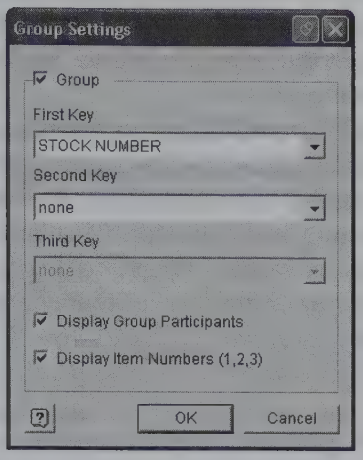


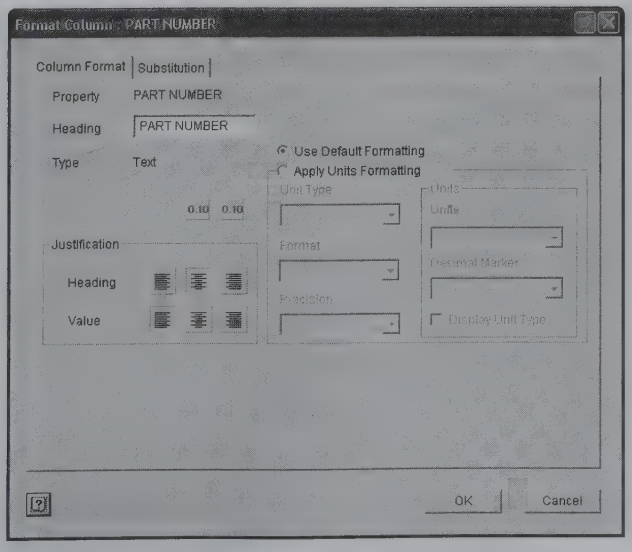
Figure 16-44.
The **Group Settings** dialog box.



identify whether or not to display group participants and item numbers by selecting or deselecting the appropriate check boxes.

- **Property.** This column shows all the properties used in the parts list. Right-clicking on any of the properties shows pop-up menu containing the following options: **Column Chooser** and **Format Column**. Pick **Column Chooser** to open the **Parts list Column Chooser** dialog box described earlier in this section. Pick the **Grouping** button to access the **Group Settings** dialog box.
  Pick **Format Column** to access the **Format Column** dialog box. See Figure 16-45. The **Format Column** dialog box contains variables that control the format of the property listed next to the **Property:** name. Use the **Heading** edit box to modify the column name. **Type** identifies the property column value category, such as text or numeric, as defined in the **Properties** dialog box. Select the **Leading Zeros** button to display leading zeros, which changes the display of .5 to 0.5, for example. Select the **Trailing Zeros** button to display trailing zeros, which changes the display of 45 to 45.0, for example.

Figure 16-45.
The **Format Column** dialog box, **Column Format** tab.



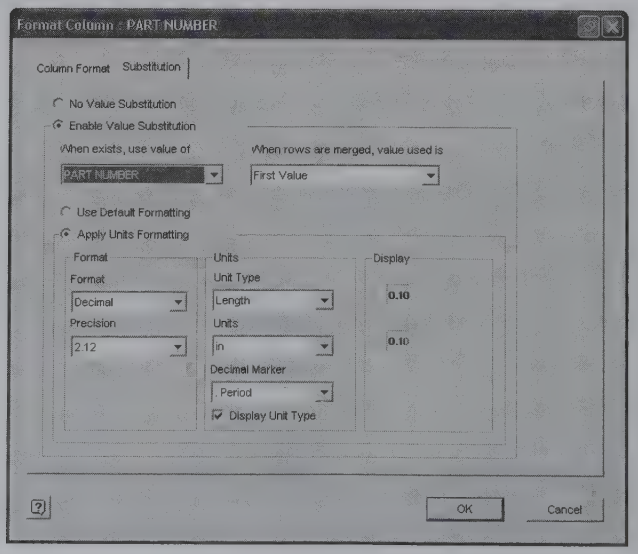
The **Justification** area includes buttons that allow you to control the left, center, or right justification, or placement of heading and value text in reference to the column cell edges. The **Units Formatting** area controls the units and display of numbers. Select **Length**, **Angularity**, **Volume**, or **Mass** from the **Unit Type** drop-down list before continuing with the other options in the **Units Formatting** area.

Pick the **Use Default Formatting** radio button to apply default unit settings to column values. The default settings are specific to the selected column property, but are defined according to the selected standard. Select the **Apply Units Formatting** radio button to override the default unit settings applied to the column values. When you pick the **Apply Units Formatting** radio button, the **Format**, **Units**, and **Display** areas become available.

Use the **Substitution** tab, shown in Figure 16-46, to replace the selected column property values with values from a different property. For example, you can choose to use Title property values in the Part Number property column. By default, property values are not substituted, and the **No Value Substitution** radio button is selected. To use different property values in the selected column, pick the **Enable Value Substitution** radio button to access the following options:

Select the **Browse Properties...** option from the **When exists, use value of** drop-down list to access the **Parts List Column Chooser** dialog box. The **Parts List Column Chooser** dialog box is used to locate or create a substitute property. The **When rows are merged, value used is** drop-down list allows you to define how values are merged, or calculated. Pick the **First Row** option to list only the value displayed in the first row when rows are merged. To calculate the values in all the column cells when rows are merged, you must select the **Sum of Values** option. Pick the **Use Default Formatting** radio button to apply default unit settings to substitute column values. The default settings are specific to the selected column property, but are typically defined according to the selected standard. Select the **Apply Units Formatting** radio button to override the default unit settings applied to the substitute column values. When you pick the **Apply Units Formatting** radio button, the **Format**, **Units**, and **Display** areas become available.

Figure 16-46.
The **Format Column** dialog box, **Substitution** tab.



The column titled **Column** identifies the name used in the parts list for each property. To change the name, highlight the name you want to modify, enter a new name, and then select outside of the cell. Use the **Width** column to define the width of each parts-list property column. To change the width, highlight the value you want to modify, enter a new width, and then select outside of the cell.

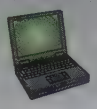
- **Comments.** Use this edit box to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

NOTE

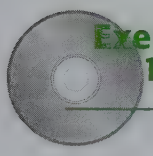


Many parts-list variables defined in the **Parts List Style** area of the **Styles and Standards Editor** can be individually overridden once a parts list is placed in the drawing.

PROFESSIONAL TIP



For most drawings, a single parts list style is all that is required.



Exercise 16-14 Complete the Exercise on the Student CD.

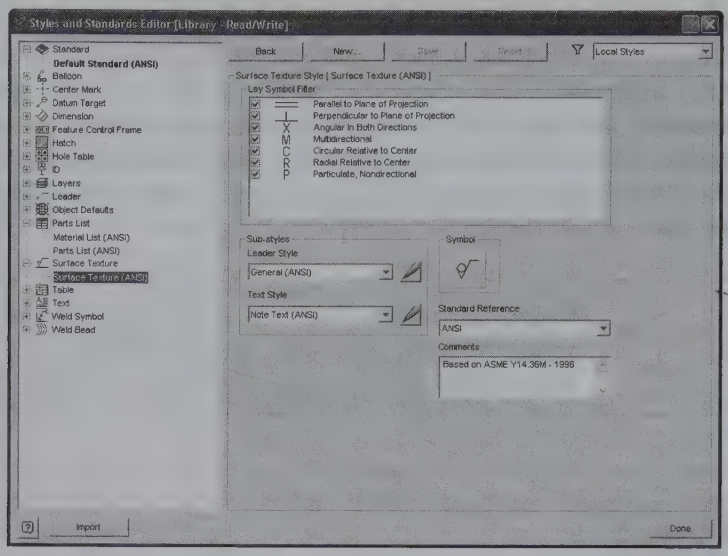
Surface Texture Styles

Surface finish refers to the allowable roughness, waviness, lay, and flaws on a surface, obtained by machining, grinding, honing, or lapping. The surface finish symbol is a "V" shape that contains values related to the finish characteristics. Surface texture styles are used to define surface finish symbol characteristics.

When you expand the **Surface Texture** node in the **Styles and Standards Editor** browser, all available surface texture styles are displayed as child nodes. Picking a specific surface texture style displays surface texture style variables, and the following areas, as shown in Figure 16-47:

- **Lay Symbol Filter.** This area contains a number of check boxes that control the machining lay symbols available to use when placing surface finish definitions. If you do not want to have access to a certain symbol when creating surface finish symbols using the **Surface Texture Symbol** dialog box, select the desired check box.
- **Sub-styles.** This area is used to select an existing leader and text style to use for surface texture symbols and contains the following options:
  - **Leader Style.** Use this drop-down list to choose the leader style attached to surface texture symbols. You can access the selected leader style variables by picking the **Edit Leader Style** button. Choosing the **Edit Leader Style** button displays the **Leader Style** area of the **Styles and Standards Editor**.
  - **Text Style.** Use this drop-down list to choose the text style used in the surface texture symbol. You can access the selected text style variables by picking the **Edit Text Style** button. Choosing the **Edit Text Style** button displays the **Text Style** area of the **Styles and Standards Editor**.

Figure 16-47.
The **Surface Texture Style** area of the **Styles and Standards Editor**.



- **Symbol.** This button allows you to set the machining prohibited symbol size when creating or modifying a custom drafting standard.
- **Standard Reference.** Use this drop-down list to select a unique drafting standard for applications when a different standard is required.
- **Comments.** Use this edit box to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

NOTE



The surface texture properties defined in the **Surface Texture Style** area of the **Styles and Standards Editor** are default. Some of the options can be overridden when you place surface texture definitions.

PROFESSIONAL TIP



For most drawings, a single surface texture style is all that is required.



Exercise 16-15 Complete the Exercise on the Student CD.

Table Styles

The drawing **Table** tool is primarily used to create tables from iPart and iAssembly parameter tables, but it can also be used to build tables that are not parametrically

controlled by model or drawing data, or for tables that cannot be made using other drawing annotation tools. Table styles define default **Table** tool settings.

When you expand the **Table** node in the **Styles and Standards Editor** browser, all available table styles are displayed as child nodes. Picking a specific table style displays table style variables, and the following areas, as shown in Figure 16-48.

- **Title.** Use this edit box to redefine the table title.
- **Line Format.** To change the line weight and color of the table border, pick the **Outside** button. Then select the desired line weight from the **Line Weight** drop-down list. Pick the **Color** button to access the **Color** dialog box, which allows you to change the table border color. To change the line weight and color of cell borders, which are the lines that separate table entries and titles, pick the **Inside** button. Then select the desired line weight from the **Line Weight** drop-down list and pick the **Color** button to access the **Color** dialog box, which again allows you to change the table border color.

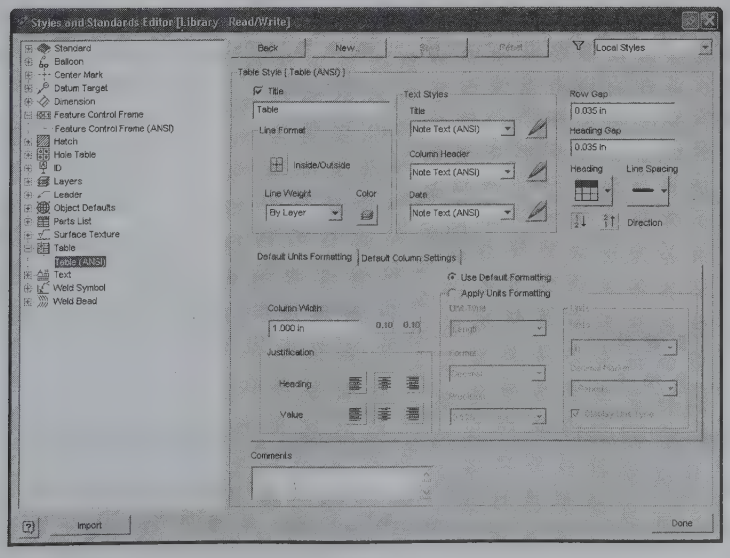
NOTE



In most situations, the **Line Weight** and **Color** variables should be set to By Layer. This will automatically assign a line weight and color to the table style based on the characteristics of the layer defined in the **Object Defaults Style** area for tables.

- **Text Styles**. This area contains drop-down lists to format the table's title, column heads, and data. Use the **Title, Column Header, and Data** drop-down lists to choose the text style of each of these table features. You can access the selected text style variables by picking the **Edit Text Style** button. Choosing the **Edit Text Style** button displays the **Text Style** area of the **Styles and Standards Editor**.
  - **Heading Gap.** Use this edit box to enter the amount of space between the parts list heading text and the heading cell.
  - **Row Gap.** Use this edit box to enter the amount of space between parts list text and cell edges.
  - **Heading.** This flyout button allows you to modify the location of the parts list heading, or completely remove the heading if necessary. By default, the **Top**

Figure 16-48.
The **Table Style** area of the **Styles and Standards Editor**.



position button is selected, which means that the parts list heading is located at the top of the table. Pick the **Bottom** position button to move the heading to the bottom of the table, or pick the **No Heading** position button to completely remove the parts list heading.

- **Line Spacing.** This flyout button allows you to select and specify single, double, or triple line spacing between multiple lines of parts list text.
- **Direction.** These buttons control the order of parts list items. Pick the **Add new parts to bottom** button to create a parts list that provides data from the bottom of the sheet to the top. Select the **Add new parts to top** button to create a parts list that provides data from the top of the sheet to the bottom.
- **Default Units Formatting.** This tab controls a variety of default table unit settings. Enter the default column width in the **Column Width** edit box. The **Justification** area includes buttons that allow you to control the left, center, or right justification, or placement of heading and value text in reference to the column cell edges. Pick the **Use Default Formatting** radio button to apply default unit settings to column values. The default settings are specific to the selected column property, but are defined according to the selected standard. Select the **Apply Units Formatting** radio button to override the default unit settings applied to the column values. When you pick the **Apply Units Formatting** radio button, the **Format**, **Units**, and **Display** areas and zero suppression buttons become available. The **Units Formatting** area controls the units and display of numbers. Select **Length**, **Angularity**, **Volume**, or **Mass** from the **Unit Type** drop-down list before continuing with the other options in the **Units Formatting** area. Select the **Leading Zeros** button to display leading zeros, which changes the display of .5 to 0.5, for example. Select the **Trailing Zeros** button to display trailing zeros, which changes the display of 45 to 45.0, for example.
- **Default Column Settings.** This tab can be used to set default table column characteristics including the number of columns, column headings, and column width. If you do not adjust any of the default column settings in this tab, column characteristics can be defined when the table is created or edited. Pick the **Add Column** button to create custom columns. The Column 1 row listed under **Property** is the first column on the left of a table, additional columns are added to the right. Pick the check box listed under **Name from Source** to use the source column name, Column 1, Column 2, and so on, by default. Deselect the check box listed under **Name from Source** to enter a name in the **Column** cell. You can also adjust column settings by right-clicking on the column you want to format in the **Property** list and picking **Format Column** to access the **Format Column** dialog box. Default columns can be deleted by selecting a row or multiple rows and picking the **Remove Column** button.
- **Comments.** Use this edit box to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

NOTE



Many table variables defined in the **Table Style** area of the **Styles and Standards Editor** can be individually overridden once a table is placed in the drawing.

Weld Symbol Styles

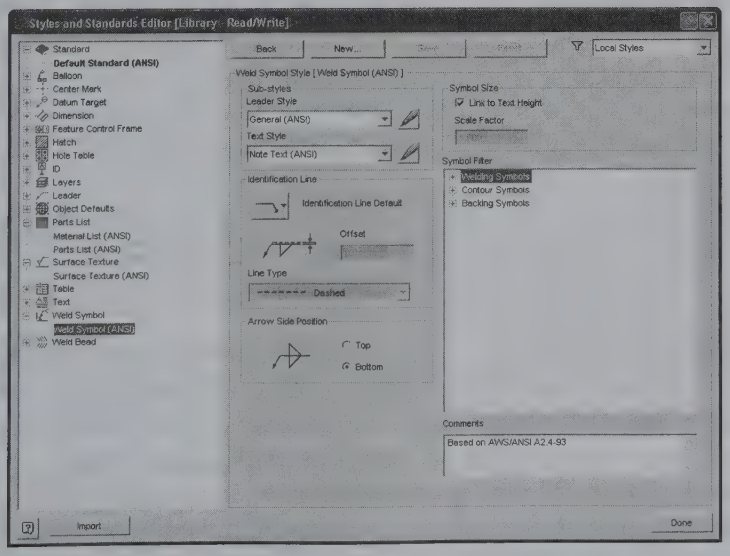
A *weldment drawing* is a two-dimensional representation of a weldment that shows how multiple parts and subassemblies are welded together. Inventor contains

tools and preferences specific to 2D weldment drawings, including weldment model symbols and annotations, end treatments, caterpillars, and drawing weld symbols. The display of caterpillars and end treatments are specified using the weld bead styles, described later in this chapter. Weld symbol and annotation display characteristics are defined using weld symbol styles.

When you expand the **Weld Symbol** node in the **Styles and Standards Editor** browser, all available weld symbol styles are displayed as child nodes. Picking a specific weld symbol style displays weld symbol style variables, and the following areas, as shown in Figure 16-49:

- **Sub-styles.** This area is used to select an existing leader and text style to use for weld symbols and contains the following options:
  - **Leader Style.** Use this drop-down list to choose the leader style attached to weld symbols. You can access the selected leader style variables by picking the **Edit Leader Style** button. Choosing the **Edit Leader Style** button displays the **Leader Style** area of the **Styles and Standards Editor**.
  - **Text Style.** Use this drop-down list to choose the text style used in the weld symbol. You can access the selected text style variables by picking the **Edit Text Style** button. Choosing the **Edit Text Style** button displays the **Text Style** area of the **Styles and Standards Editor** previously described.
- **Symbol Size.** This area is used to define the size of weld symbols. You can choose to size the symbol according to the height of the text by selecting the **Link to Text Height** check box. Or, if you do not select the **Scale to Text Height** check box, the **Scale Factor** edit box becomes available, allowing you to enter a scale factor between .5 and 2, such as 1.5 which makes weld symbols 1.5 times the size of the default value.
- **Identification Line.** This area includes the following identification line preferences:
  - **Identification Line Default.** This flyout button is used to control the placement of the identification line, if used. The **No Identification Line** button is selected by default. Pick the **Identification Line Above** or **Identification Line Below** button, depending on the application.

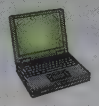
Figure 16-49.
The **Weld Symbol Style** area of the **Style and Standards Editor**.



- **Offset.** Enter the amount of space between the welding symbol identification line and the welding symbol using this edit box.
- **Line Type.** Use this drop-down list to change the default welding symbol identification line type.

- **Symbol Filter.** This area contains a tree structure with a Welding Symbols, Contour Symbols, and Backing Symbols node. Expanding each parent node displays child nodes, check boxes, and radio buttons that control the symbols available when placing drawing weld definitions. If you want to have access to only certain weld, contour, or backing symbols, navigate through the tree structure and select or deselect the check box or radio button that corresponds to the symbol to which you do or do not want to have access.
- **Arrow Side Position.** Pick the **Top** radio button to place the arrow side symbol above the weld symbol by default, or select the **Bottom** radio button to place the arrow side symbol below the weld symbol by default.
- **Comments.** Use this edit box to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

PROFESSIONAL TIP



For most drawings, a single weld symbol style is all that is required.



Exercise 16-16 Complete the Exercise on the Student CD.

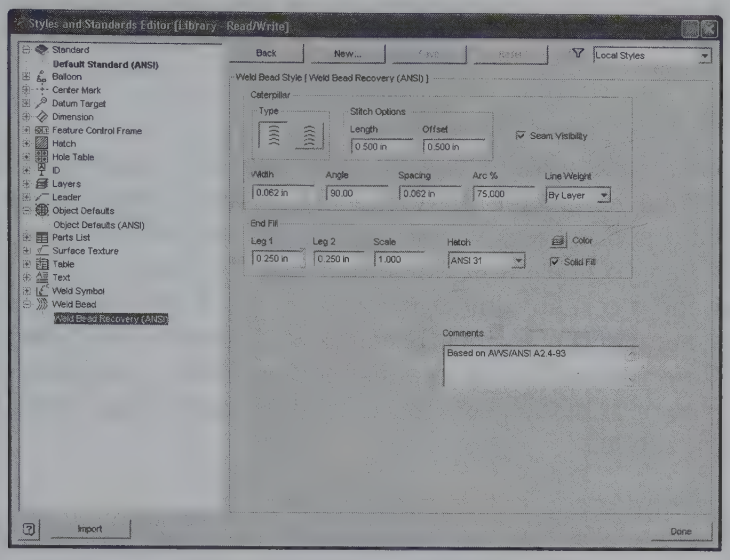
Weld Bead Styles

The display of weldment drawing caterpillars and end treatments are specified using weld bead styles. When you expand the **Weld Bead** node in the **Styles and Standards Editor** browser, all available weld bead styles are displayed as child nodes. Picking a specific weld bead style displays weld bead style variables within the **Caterpillar**, **End Fill**, and **Comments** areas, as shown in Figure 16-50.

The **Caterpillar** area allows you to define the default weld bead symbol display characteristics applied when using the **Weld Caterpillars** dialog box, and contains the following options:

- **Type.** This area contains a **Partial** button that allows you to place a caterpillar next to an edge by default, such as with a fillet weld, and a **Full** button that allows you to place a caterpillar along both sides of an edge by default, such as with a V- or square-groove weld.
- **Stitch Options.** Use this area to specify the default characteristics of intermittent weld caterpillars. Enter the length of each weld bead in the **Length** edit box. Then, enter the center-to-center distance between weld lengths, or pitch, in the **Offset** edit box. For example, a caterpillar with a weld length of 25 mm and an offset of 45 mm has 20 mm space between weld beads.
- **Seam Visibility.** Pick this check box to display the edge on which the caterpillar is placed by default, when the caterpillar is generated.
- **Width.** Use this edit box to specify the default weld bead width.

Figure 16-50.
The **Weld Bead Style** area of the **Styles and Standards Editor**.



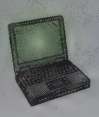
- **Angle.** This edit box is used to define the default angle between the selected edge and the caterpillar geometry. For example, you might use a 90° angle for a square groove weld and a 45° angle for a certain type of fillet weld.
- **Spacing.** Use this edit box to define the default spacing between caterpillar arcs.
- **Arc %.** This edit box is used to specify the default radius of the caterpillar arc in reference to the arc chord, or bead width.
- **Line Weight.** Use this drop-down list to select the default caterpillar line weight. In most situations, the **Line Weight** variable should be set to By Layer, which will automatically assign a line weight to the weld bead style based on the characteristics of the layer defined in the **Object Defaults Style** area for end fills.

The **End Fill** area allows you to define the default settings of the **End Fill** dialog box and contains the following options:

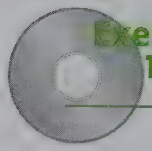
- **Leg 1.** Use this edit box to enter the default size of the first leg of Bevel, J-Type, V-Type, and U-Type end fills.
- **Leg 2.** Use this edit box to enter the default size of the second leg of Bevel, J-Type, V-Type, and U-Type end fills.
- **Scale.** Use this edit box to specify the default hatch pattern scale setting by entering a scale, or pick the flyout button and select a scale.
- **Hatch.** Use this drop-down list to choose the default end fill hatch pattern.
- **Color.** Pick this button to access the **Color** dialog box to identify the default fill color. In most situations, the **Color** variable should be set to By Layer, which will automatically assign a color to the end fill style based on the characteristics of the layer defined in the **Object Defaults Style** area for end fills.
- **Solid Fill.** Pick this check box to specify the default end fill option as a solid color.

Use the **Comments** edit box to add information about the selected style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

PROFESSIONAL TIP



For most drawings, a single weld bead style is all that is required.



Exercise 16-17 Complete the Exercise on the Student CD.

Working with Object Defaults

When you create a drawing using Inventor, drawing geometry and annotations are created according to specific predefined style and layer settings. For example, when you place a drawing view, visible object lines are drawn using a visible object line layer, and when you add a linear dimension, the dimension is drawn using a dimension layer and a certain dimension style. Every drawing item, such as hidden lines, centerlines, angular dimensions, border, and title block are all assigned a specific object default. ***Object defaults*** are responsible for linking specific object styles and layers to every type of drawing geometry and annotation. Object defaults are defined using object default styles.

NOTE

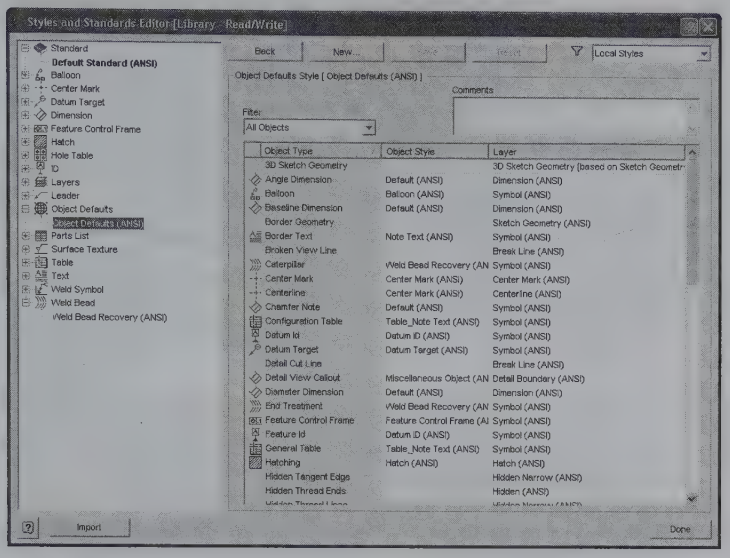


Object defaults are intended to describe the default styles and layers used when creating a drawing. Using object defaults is very effective for producing drawings, because drawing geometry and annotations can be drawn once, without modification. However, most object defaults can be overridden if absolutely necessary once drawing geometry and annotations are added.

When you expand the **Object Default** node in the **Styles and Standards Editor** browser, all available object default styles are displayed as child nodes. Picking a specific object style displays object default style variables. See Figure 16-51. The **Object Default Style** area contains the following three primary columns.

- **Object Type.** This column lists all possible objects that you can create in the drawing environment. For example, when you add a drawing view, visible object lines are created according to the **Visible View Edge** object type. When you place a linear dimension, the dimension is created according to the **Linear Dimension** object type. The purpose of object types is to correlate the drawing objects to the appropriate style and layer. You cannot modify object types, as they are an element of the Inventor program. However, you can adjust each object type style and layer based on specific industry or company standards. Object type characteristics are set using the **Object Style** and **Layer** columns.
- **Object Style.** This column contains a drop-down list for all object types that require a style designation. Some object types reference an object style and a layer, while other object types only reference a layer. For example, the **Diameter Dimension** object type uses a dimension style, while the **Hidden View Edge** object type only uses a layer. Object styles include the leader, text, balloon, center mark, datum target, dimension, feature control frame, hatch, hole table, ID, parts list, surface texture, weld symbol, and weld bead styles. However, only specific styles are offered from each drop-down list. For example, only dimension styles are listed in the **Object Style** drop-down list of the **Angle Dimension** object type. Use the **Object Style** drop-down lists to select the style applied to each object type.
- **Layer.** This column contains a drop-down list for all object types, and is used to select the layers applied to each object type. Layers are defined using the **Layer Styles** area of the **Styles and Standards Editor**. Pick the desired layer from the

Figure 16-51.
The **Object Default Style** area of the **Styles and Standards Editor**.



Layer drop-down list for each object type. For example, the Border (ANSI) layer might be used for the **Border Geometry** object type, and the Dimension (ANSI) layer might be used for **Linear Dimension** object type.

All object types can be displayed by selecting the **All Objects** option of the **Filter** drop-down list as shown in Figure 16-51. Displaying all object types can be confusing. To display, or filter, only a specific group of object types, select **Dimension Objects**, **Leader Objects**, **Model/View Objects**, or **Text Objects** from the **Filter** drop-down list. Additionally, the **Comments** edit box is available, and can be used to add information about the selected object default style. A comment can include content about variables used to define the style, the standard with which the style is associated, or any other data that can help clarify the style settings.

NOTE

You must select the desired object default style in the **Standard** area, **General** tab of the **Styles and Standards Editor**, in order for the correct object defaults to be applied.

Exercise 16-18

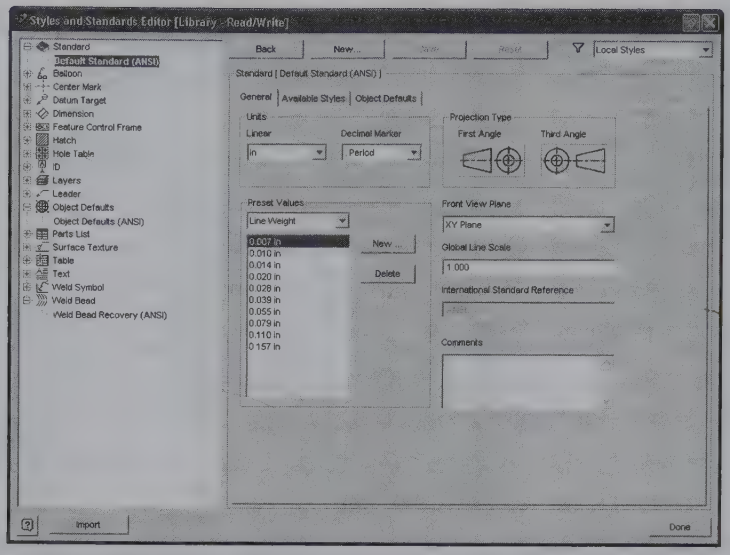
Complete the Exercise on the Student CD.

Working with Standards

A ***standard*** contains selected styles from each style type, and also defines several other general drawing preferences. The ***active standard*** specifies which styles are applied to a drawing. A standard is used to define exactly how a drawing looks according to selected styles, and style variables.

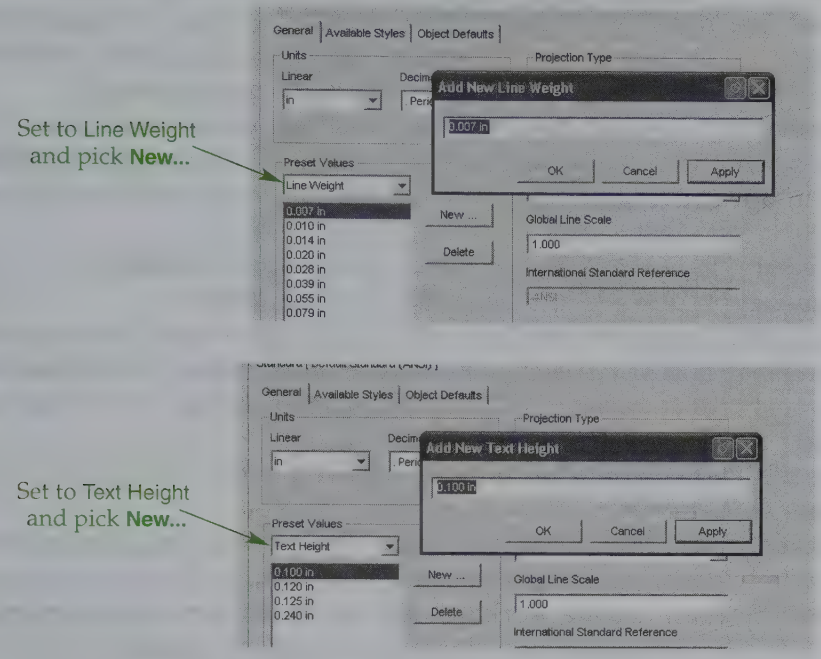
When you expand the **Standard** node in the **Styles and Standards Editor** browser, all available standards are displayed as child nodes. Picking a specific standard displays standard variables, and the following tabs, as shown in Figure 16-52:

Figure 16-52.
The **Standard** area of the **Styles and Standards Editor**.



- **General.** This tab includes the following general standard settings:
  - **Units.** Use this area to set the overall, or common, drawing units. Specify the type of unit measurement such as inch or mm from the **Linear** drop-down list. You can also define the type of decimal marker, such as a decimal point or comma, from the **Decimal Marker** drop-down list.
  - **Projection Type.** This area allows you to specify drawing view projection angles. Pick the **First angle of projection** button to project first angle, or pick the **Third angle of projection** button to project third angle.
  - **Preset Values.** Use this area to identify the line weight and text height values available in **Line Weight** and **Text Height** or **Text Size** drop-down lists found in many dialog boxes, and **Styles and Standards Editor** style areas. Pick **Line Weight** from the drop-down list to display the line weights currently available. To add another line weight, pick the **New...** button, enter a value in the **Add New Line Weight** dialog box and pick the **Apply** button to add the new line weight. See Figure 16-53. If you want to delete a line weight, select the one you want to remove and pick the **Delete** button. Select **Text Height** from the drop-down list to display the text heights currently available. To add another text height, pick the **New...** button, enter a value in the **Add New Text Height** dialog box and pick the **Apply** button to add the new text height. If you want to delete a text height, select the one you want to remove and pick the **Delete** button.
  - **Global Line Scale**. Use this edit box to modify the line scale factor applied to all existing line styles.
  - **International Standard Reference.** This display box identifies the drafting standard associated with the drawing standard.
  - **Active Object Defaults.** Use this drop-down list to choose the object default style used in the standard. You can access the selected object default variables by picking the **Edit Object Default Style** button. Choosing the **Edit Object Default Style** button displays the **Object Default** area of the **Styles and Standards Editor**.
  - **Comments.** Use this edit box to add information about the selected standard. A comment can include content about variables used to define the standard, or any other data that can help clarify the standard settings.

Figure 16-53.
The **Add New Line Weight** dialog box and the **Add New Text Height** dialog box are available in the **Preset Values** area.



- **Available Styles.** This tab controls, which styles are available and displayed in the **Styles and Standards Editor** when the **Active Standard** filter option is set. Pick the style from the **Style Type** list box, and then select the check boxes in the **Name** column that correspond to the styles you want associated with the active standard.

PROFESSIONAL TIP



The **Available Styles** tab of the **Standard** area functions like a filter operation, and allows you to display and have access to only specific styles related to the active standard when the **Active Standard** filter option is set. For example, if you only use one style for a particular style type, that should be the only style selected.



Exercise 16-19 Complete the Exercise on the Student CD.

Controlling Styles and Standards

Once you create or modify styles and standards, you must update, save, and purge the styles and standards from the styles library to ensure that styles and standards are associated with the styles library and to avoid confusion. Refer to Chapter 3 for more information on updating, saving, and purging styles and standards from the styles library.

Document Settings

The **Document Settings** dialog box controls several work environment options associated with the current file. You should be familiar with many of these settings before you create a drawing. Additionally, document settings can be customized in template files so that specific options are already set each time you start a new drawing.

To access document settings and the **Document Settings** dialog box, select the **Document Settings...** option from the **Tools** pull-down menu. The **Document Settings** dialog box available in a drawing file, and shown in Figure 16-54, includes the following tabs:

- **Standard.** This tab, shown in Figure 16-54, contains the **Active Standard** drop-down list, which is used to activate a specific drawing standard.
- **Sketch.** You should already be familiar with this tab, shown in Figure 16-55, which contains snap and grip spacing options for 2D sketches. 2D sketches are used to generate borders, title blocks, draft views and symbols in the drawing environment.
- **Drawing.** This tab, shown in Figure 16-56, allows you to control the following drawing document settings:
  - **Defer Updates.** Select this check box if you do not want to automatically update the drawing when changes are made to the model. Deferring updates may be helpful when you are working with large, complex drawing views that take a long time to regenerate after model geometry changes. Deselect the **Defer Updates** check box if you want updates to occur automatically.

NOTE



You can also defer the update of a specific drawing file before you open the drawing. To use this method, first access the **Open** dialog box. Then select the drawing file you want to open, followed by picking the **Options...** button to display the **File Open Options** dialog box. Select the **Defer Updates** check box and pick **OK** button to defer updates.

Figure 16-54.
The **Document Settings** dialog box, **Standard** tab.

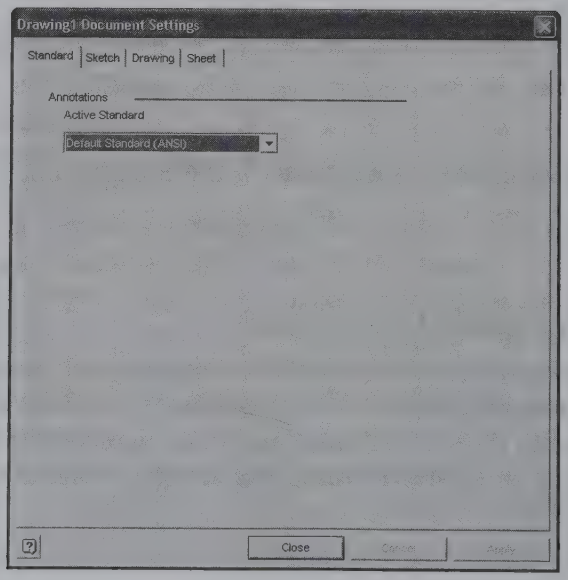


Figure 16-55.
The **Document Settings** dialog box, **Sketch** tab.

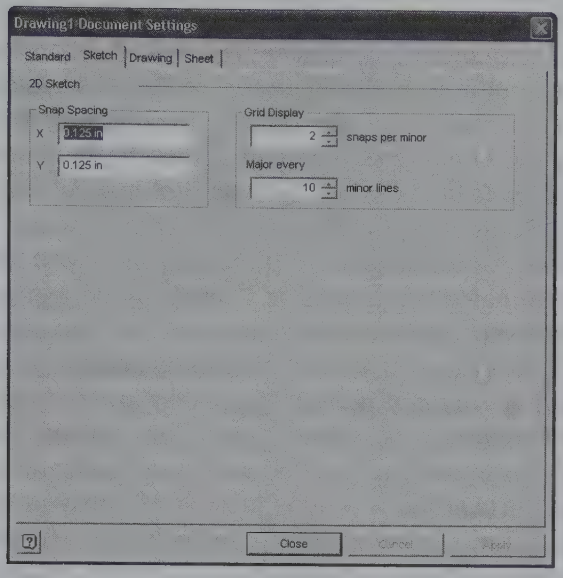
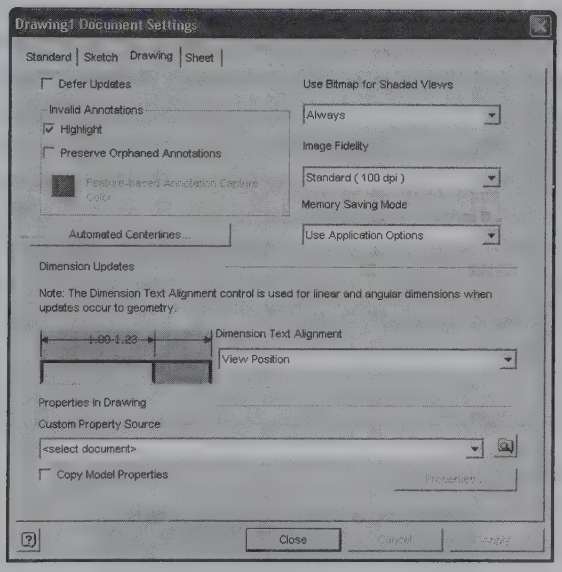


Figure 16-56.
The **Document Settings** dialog box, **Drawing** tab.



- **Invalid Annotations.** This area contains options for dealing with invalid drawing annotations, such as those moved away from a drawing view associated with model geometry, known as ***orphaned annotations***. Pick the **Highlight** check box to highlight any invalid annotations placed in the active drawing. Deselect this check box if you want to display invalid dimensions without highlighting, or when inserting DWG files into a drawing. Pick the **Preserved Orphaned Annotations** check box to keep orphaned annotations in the drawing. When the **Preserved Orphaned Annotations** check box is selected, you can pick the **Feature-based Annotation Capture Color** button to choose a color for invalid annotations.

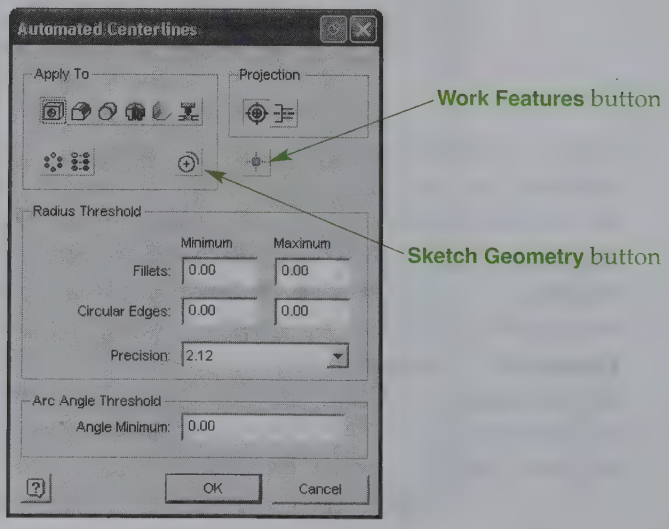
- **Automated Centerline...** Pick this button to access the **Centerline Settings** dialog box shown in Figure 16-57. The **Centerline Settings** dialog box is used to specify which drawing items automatically receive centerlines and center marks when a drawing view is placed.

  The **Apply To** area contains a number of buttons that control the types of drawing geometry that will automatically receive centerlines and center marks. To define automated centerlines, pick the buttons in the **Apply To** area that correspond with the drawing items to which you automatically want centerlines or center marks added. For example, pick the **Hole Features** button to add centerlines and center marks to drawing views that contain hole features.

  The **Projection** area contains two buttons that allow you to further identify when you want automated centerlines and center marks placed. Pick the **Objects in View, Axis Normal** button to add center marks to circular or radial features that are normal to the view plane, and pick the **Objects in View, Axis Parallel** button to add centerlines to longitudinal features that are parallel to the view plane.

  The **Radius Threshold** area allows you to limit the number of centerlines and center marks automatically added to drawing views, based on the selected **Apply To** and **Projection** buttons. You may first want to set the precision of the threshold limits by selecting a precision from the **Precision** drop-down list. The precision defines the precision, or number, of decimal places you can enter in the **Fillets:** and **Circular Edges:**, **Minimum** and **Maximum** edit boxes. For example, if you select a precision of 3.123, the value you enter in the edit boxes will round to the nearest thousandth of a value, such as .063 instead of .0625. Use the **Fillets: Minimum** and **Maximum** edit boxes to specify a minimum and maximum size value to which automated centerlines and center marks are added to fillet features. For example, if you specify a minimum fillet value of .125" and a maximum fillet value of 5", only fillets that are greater than or equal to .125", or less than or equal to 5", will automatically receive centerlines and center marks. Use the **Circular Edges: Minimum** and **Maximum** edit boxes to specify a minimum and maximum size value to which automated centerlines and center marks are added to circular or radial features. For example, if you specify a minimum circular edge value of .125" and a maximum fillet value of 5", only circular or radial geometry greater than or equal to .125", or less than or equal to 5", will automatically receive centerlines and center marks.

Figure 16-57.
The **Automated Centerlines** dialog box.



Use the **Arc Angle Threshold** area to limit the number of centerlines and center marks automatically added to drawing views, based on the selected **Apply To** and **Projection** buttons. The **Arc Minimum:** edit box is used to specify a minimum size value to which automated centerlines and center marks are added to arcs, circles, and ellipses.

Once you fully define the automated centerline settings, pick the **OK** button to close the dialog box.

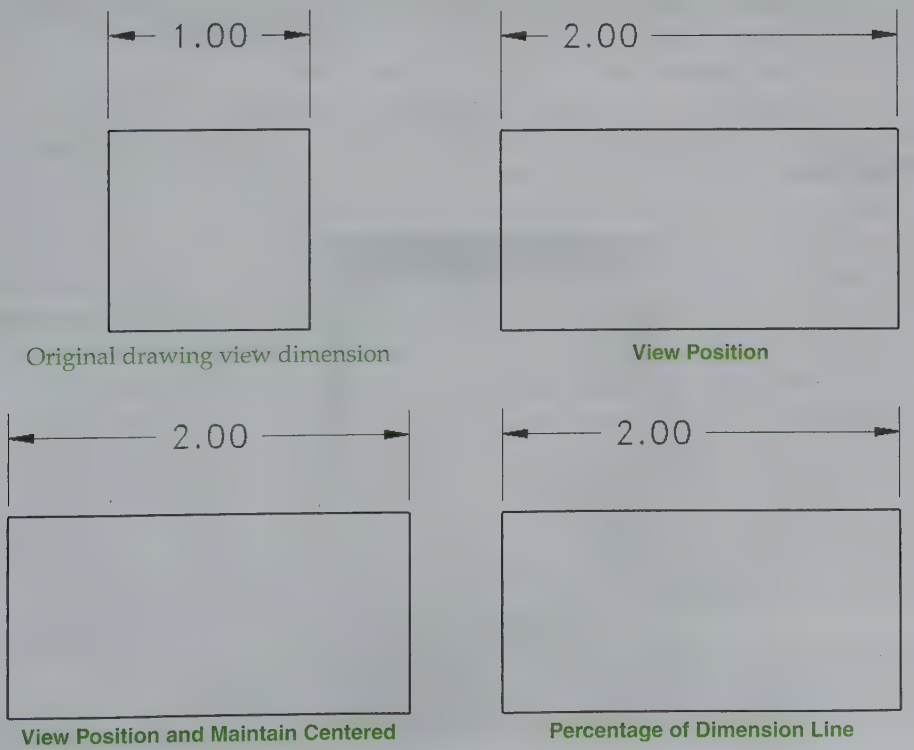
NOTE



You can also set automated centerlines and center marks for individual drawing views once they are added to the drawing sheet.

- **Use Bitmap for Shaded Views.** Pick **Always** or **Offline Only** from this drop-down list to control when bitmap images are used in shaded views. Selecting **Always** results in optimum system performance.
- **Memory Saving Mode.** Choose the desired memory saving mode from this drop-down list. Selecting **Use Application Options** sets the system performance and capacity according to the options set in the **Application Options, Drawing** tab. Pick the **Always** option to increase system performance, but limit some drawing functions, such as **Undo** and **Redo**. Select the **Never** option to disable all memory-saving operations.
- **Image Fidelity.** This drop-down list allows you to select the print quality, from 50 dots per inch to 200 dots per inch, when printing a shaded drawing view with the **Defer Updates** check box selected.
- **Dimension Text Alignment.** Use this drop-down list to set the default position of dimension text when dimensioned features are modified. See Figure 16-58. Select the **View Position** option to keep dimension text in

Figure 16-58.
The effects of using various dimension text alignment options.



its original location when a dimension value is modified. Pick the **View Position and Maintain Centered** option to center the dimension text between the extension lines. This option will only center the dimension value if the original dimension value was centered. If the original dimension text was not centered, the modified text will remain in the original dimension text location. Choose the **Percentage of Dimension Line** option to move the dimension text a percentage of the new distance between extension lines.

- **Properties in Drawing.** This area is used to specify which iProperties are copied from the **Properties** dialog box of a model to the iProperties of the drawing when the first drawing view is added to the sheet. If you created custom iProperties in a certain file, these properties can be used in the active drawing file by picking the **Browse** button and selecting the file with the custom properties using the **Open** dialog box. If you choose not to add custom properties from the selected file, pick the **<None>** option from the **Custom Property Source** drop-down list. When the first drawing view is added to a sheet, you can copy the iProperties from the model used to create the view into the iProperties of the active drawing. To accomplish this task, pick the **Copy Model Properties** check box, and then pick the **Properties...** button. Selecting the **Properties...** button opens the **Properties** dialog box. See Figure 16-59. Use the **Properties** dialog box to select individual properties to copy from the model to the drawing, or pick the **All Properties** check box to copy all of the model properties to the drawing file. When finished, pick the **OK** button to exit the dialog box.

- **Sheet.** This tab, shown in Figure 16-60, allows you to adjust sheet labels and sheet display colors. ***Sheet labels*** define the names of drawing elements displayed in the browser. By default, the sheet label is Sheet, the view label is VIEW, and the draft view label is DRAFT. See Figure 16-61. The **Colors** area allows you to define the colors used in the drawing environment. **Sheet** sets the color of the sheet, or "paper", **Sheet Outline** specifies the color of the sheet edges, and **Highlight** sets the color of drawing elements that can be selected (this color is visible when you move your cursor over selectable objects). Finally, **Selection** specifies the color of drawing elements that have been selected and are currently active. Each of the drawing element colors can be modified by picking the **Color** button next to the element. When you pick the **Color** button, the **Color** dialog box appears, allowing you to select the desired line color.

Figure 16-59.
The **Properties** dialog box.

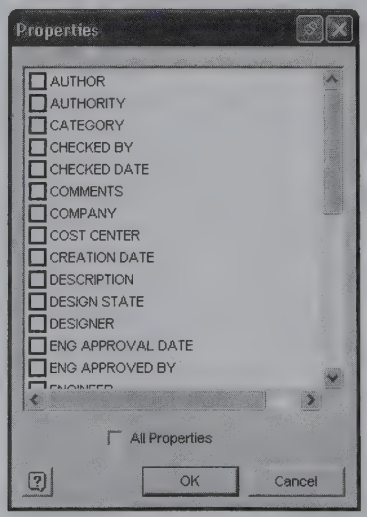


Figure 16-60.
The **Document Settings** dialog box, **Sheet** tab.

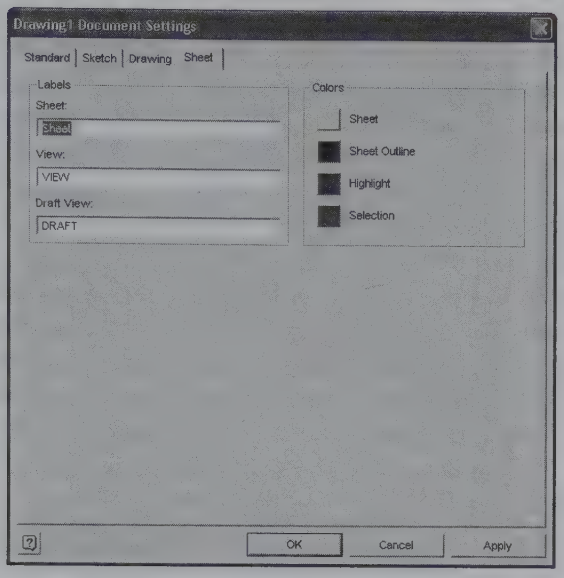
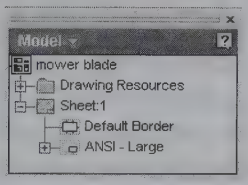


Figure 16-61.
Drawing element names in the browser.

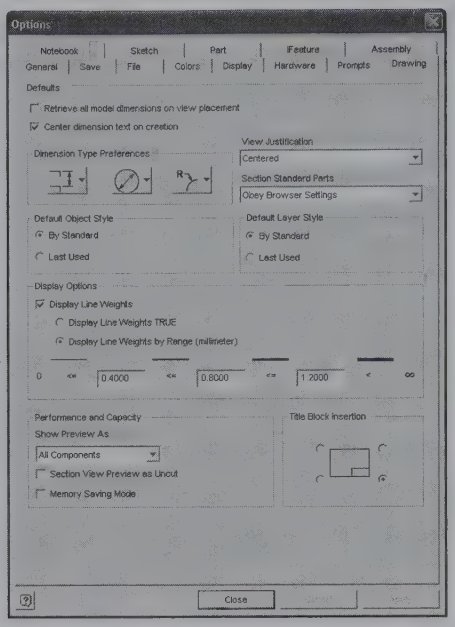


Application Options

Inventor application options are contained in the **Options** dialog box, which you access by picking the **Application Options...** selection of the **Tools** pull-down menu. Unlike document settings, Inventor application options are general program options, and are not specific to any particular file. However, many application options do help configure work environments. You have already explored many **Application Options** dialog box tabs. Though application options are not set in templates, because they apply to the entire program, you should become familiar with the following options in the **Drawing** tab, shown in Figure 16-62, before you begin work on a drawing:

- **Retrieve all model dimensions on view placement.** If this check box is selected, when you insert the view model, dimensions are automatically placed. You can override this setting when you create a view using the **Base View** tool.
- **Center dimension text on creation.** Pick this check box to automatically center model dimension text between extension lines. If this check box is not selected, model dimension text is placed according to their position in the model.
- **Dimension Type Preferences.** This area includes the following three flyout buttons that can be used to define the default dimensioning format used when applying linear, diameter, and radial dimensions.
  - **Linear.** Pick the **Linear Dimension**, **Linear Diametric**, or **Linear Symmetric** from this flyout button depending on the application. Most linear dimensions should be applied using the **Linear Dimension** button.

Figure 16-62.
The **Options** dialog box, **Drawing** tab.



- **Circle.** This flyout button contains a **Diametric Dimension** and **Radial Dimension** button, though most circles are dimensioned with a diameter dimension, using the **Diametric Dimension** button.
- **Arc.** Depending on the application, most arcs are dimensioned using the **Radial Dimension** option of this flyout button. However, you can choose the **Diametric Dimension**, **Arc Length**, or **Chord Length** button if necessary.

- **View Justification.** Use this drop-down list to set the default position of drawing views.
- **Section Standard Parts.** This drop-down list allows you to define the sectioning preference of standard parts in an assembly drawing. Typically, standard assembly parts, like a bolt, are not sectioned in an assembly section view. If this is the case with your drawing, select the **Never** option from the **Section Standard Parts** drop-down list. However, if you do want to section standard parts in an assembly section view, pick the **Always** option. Another option is to pick **Obey Browser Settings**, which bases the sectioning preference of standard parts on the browser setting. The default browser setting displays standard parts as not sectioned.
- **Default Object Style** and **Default Layer Style.** Select the **By Standard** or **Last Used** radio button from these areas to define the type of object and layer styles to use when drawings are reopened. Pick the **By Standard** radio button to set object and layer styles according to the active standard object defaults. Pick the **Last Used** radio button to use the object and layer styles last used when the drawing file was last open.
- **Display Options.** This area contains options for specifying the display of drawing object line weights. If the **Display Line Weights** check box is not selected, all lines are displayed with the same line weight. To show different line weights, pick the **Display Line Weights** check box, then select one of the following radio buttons:
  - **Display Line Weights TRUE.** Pick this button to display line weights as they are specified in the active standard.

- **Display Line Weights by Range (millimeter).** Select this option to enter a range of values used to display line weights. Line weights from 0 to greater than or equal the value you enter in the left edit box are displayed thin, line weights from the value you enter in the left edit box to greater than or equal the value you enter in the center edit box are displayed thicker, line weights from the value you enter in the center edit box to greater than or equal the value you enter in the right edit box are displayed thicker, and line weights from the value you enter in the right edit box to greater than infinity are displayed thickest.

- **Performance and Capacity.** This area contains options for increasing the performance and capacity of your system. For example, picking **All Components** from the **Show Preview as** drop-down list shows a complete preview when new drawing views are created, while choosing **Boundary Box** only shows the view boundary to increase system performance.
- **Title Block Insertion.** Use this area to define the location of the drawing title block. Select the radio button that corresponds to the corner where you want the title block to be inserted.

When you are satisfied with the settings applied in the **Options** dialog box, pick the **Apply** button to apply the changes, and the **OK** button to exit the dialog box.

Drawing Sheet Parameters

Depending on your approach, the first step in developing a drawing is to define the sheet parameters, including the sheet name, size, and layout. The sheet size is the physical limits of the sheet, or the size of the paper. When you open a new Inventor drawing file for the first time, a sheet named Sheet:1 is available. The size of the sheet depends on the template you open. For example, if you open ANSI [in].idw, the default sheet size is C, or 17″ × 22″. If you want to use the default sheet parameters, you do not have to make any modifications and can continue to the next drawing phase. However, if you want to redefine the current sheet information, right-click the existing sheet in the browser and select the **Edit Sheet...** menu option to access the **Edit Sheet** dialog box. See Figure 16-63.

The following options are available inside the **Edit Sheet** dialog box:

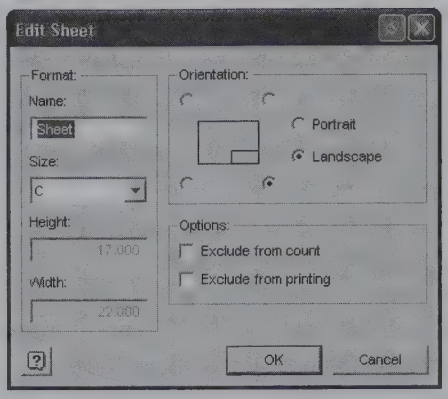
- **Name.** This edit box allows you to rename the sheet by highlighting the current name and typing a new name. The name you enter is followed by a colon and the sheet number. For example, if the sheet you are editing is the first sheet in the drawing, the name is followed by :1.
- **Size.** This drop-down list allows you to specify the sheet size. There are a number of standard sheet sizes available such as A, B, C, D, E, and F, or you can define your own sheet size by selecting the **Custom Size** option for the desired units such as **Custom Size (inches)** or **Custom Size (mm)**. Once you choose a custom size option, the **Height** and **Width** edit boxes become active, allowing you to enter the desired height and width of the custom sheet size.
- **Orientation.** This area consists of four radio buttons that correspond to the corner where the title block is placed. To change the orientation of the title block, pick the desired radio button. A preview of the action is displayed.

NOTE



The title block orientation radio buttons are available only when there is an existing title block on the sheet.

Figure 16-63.
The **Edit Sheet** dialog box.



- **Portrait.** This radio button allows you to apply a portrait, or vertical, sheet orientation.
- **Landscape.** This radio button allows you to apply a landscape, or horizontal, sheet orientation.
- **Exclude from count.** When this check box is selected, the sheet is excluded from the count of sheets in the drawing. A sheet that is excluded from the count does not have a number attached to its name.

PROFESSIONAL TIP



If only one sheet is present, the sheet number is always 1, unless you exclude the sheet from the count. If more than one sheet is present in the drawing, you can drag sheets above or below other sheets to modify the sheet number.

- **Exclude from printing.** When this check box is selected, the sheet is excluded from the printing process and is not printed with the other drawing sheets.

Once you modify the desired options, pick the **OK** button to complete the sheet edit.

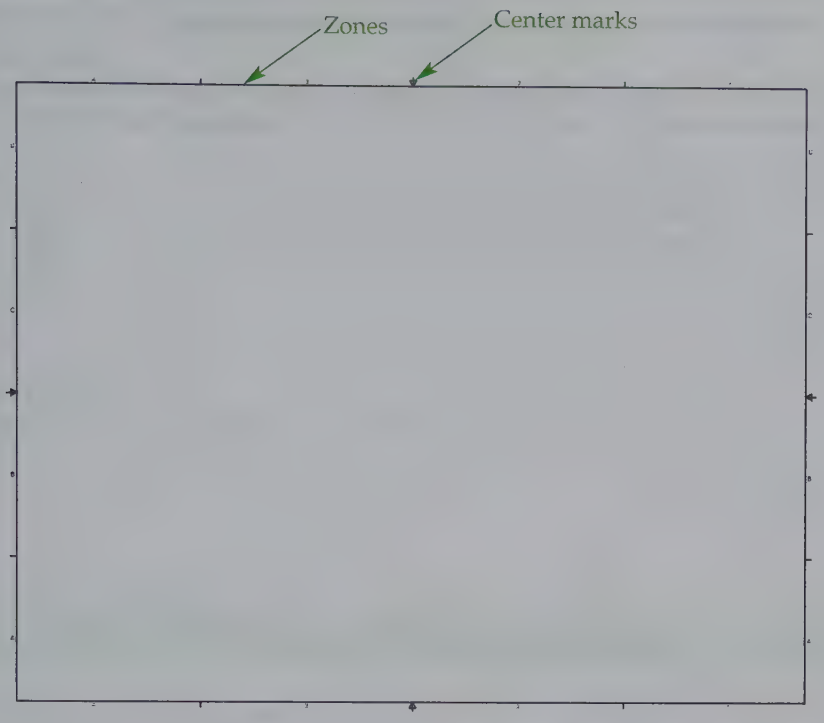


Exercise 16-20 Complete the Exercise on the Student CD.

Using Default Borders

Once you define the sheet name, size, orientation, and other parameters, depending on your approach, the next step in developing a drawing is to define a border. A border can be as simple as a rectangle inside the sheet limits, or it may include zone numbers and center marks. See Figure 16-64. When you open an Inventor drawing file for the first time, a border is already placed on the sheet. To redefine the current border on Sheet:1, you must first remove the existing border by right-clicking the default border and selecting the **Delete** menu option, or by picking the default border and pressing the [Delete] key. Next, access the **Default Drawing Border Parameters** dialog box, shown in Figure 16-65, by right-clicking on the **Borders** folder of the **Drawing**

Figure 16-64.
An example of a drawing border.



Resources folder, and selecting the **Define New Zone Border...** option, right-clicking on the **Default Border** located in the **Borders** folder of the **Drawing Resources** folder, and selecting the **Insert Drawing Border...** option, or selecting the **Define New Zone Border...** option from the **Format** pull-down menu.

The **Default Drawing Border Parameters** dialog box contains the following options that allow you to create a specific border. Refer to **Figure 16-66** for further information regarding default border options:

Figure 16-65.
The **Default Drawing Border Parameters** dialog box.

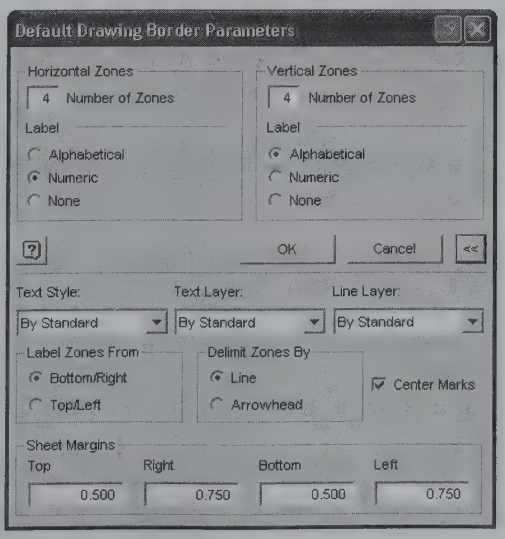
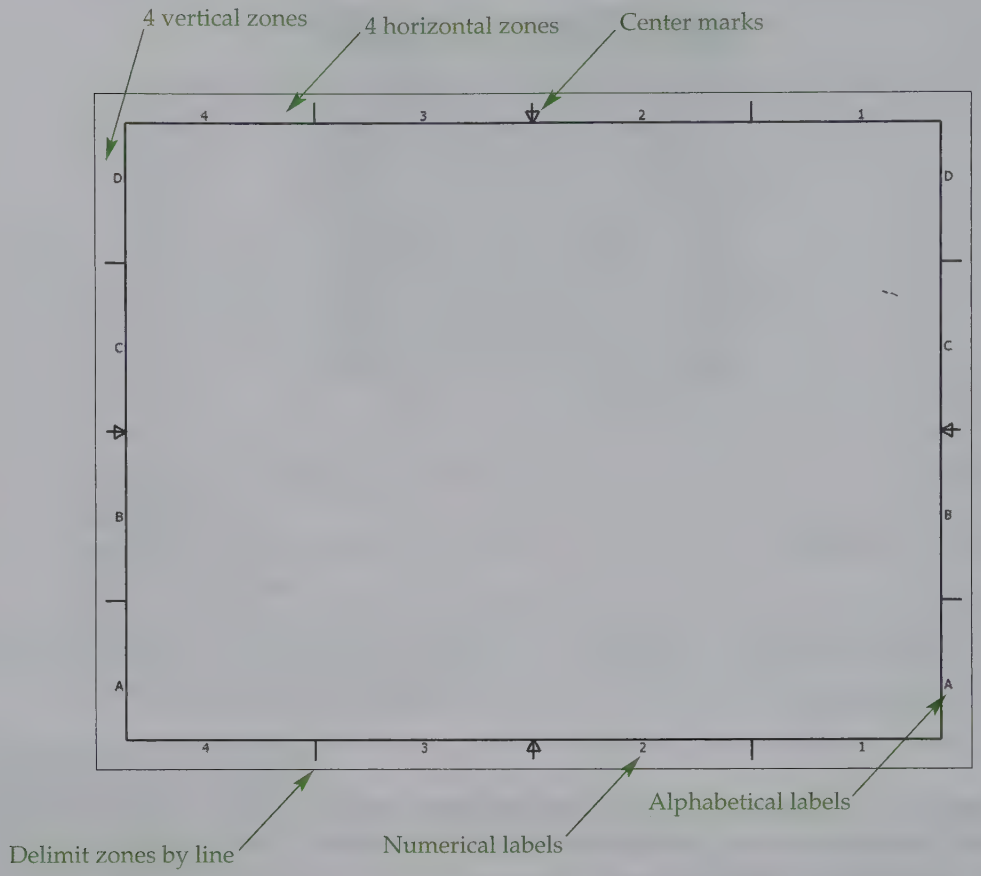


Figure 16-66.
Default border parameters, label zones begin from the bottom-right corner.



- **Horizontal Zones.** This area allows you to define the horizontal border zones applied to the top and bottom of the border. Specify the number of zones you want to create in the **Number of Zones** edit box. Then pick the type of zone label by selecting the **Alphabetical** radio button to use letters, or the **Numerical** radio button to use numbers. You can also suppress horizontal zones by choosing the **None** radio button.
- **Vertical Zones.** This area allows you to define the vertical border zones applied to the left and right sides of the border. Specify the number of zones you want to create in the **Number of Zones** edit box. Then pick the type of zone label by selecting the **Alphabetical** radio button to use letters, or the **Numerical** radio button to use numbers. You can also suppress vertical zones by choosing the **None** radio button.
- **Text Style.** Use this drop-down list to specify the text style for border text.
- **Text Layer.** Use this drop-down list to indicate the layer assigned to border text.
- **Line Layer.** Use this drop-down list to identify the layer assigned to border geometry.
- **Label Zones From.** This area is used to define zone label justification. Pick the **Bottom/Right** radio button to label the zones from the bottom/right corner of the border, or pick the **Top/Left** radio button to label the zones from the top/left corner of the border.
- **Delimit Zones By.** This area allows you to specify how you want to mark zone limits. Pick the **Line** radio button to mark the zone limits with small lines, or pick the **Arrowhead** radio button to mark the zone limits with an arrowhead.

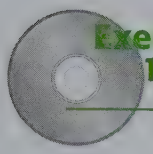
- **Center Marks.** By default, this check box is selected. If you do not want to display center marks, deselect the **Center Marks** check box.
- **Sheet Margins.** This area is used to define the top, right, bottom, and left border margins. The margins define the placement of the border in reference to the sheet edges.

Once you fully define the default border parameters, pick the **OK** button to insert the border onto the current sheet.

NOTE



A default border can also be added to a drawing by pressing [Ctrl] + [Z] keys on your keyboard.



Exercise 16-21 Complete the Exercise on the Student CD.

Creating a Custom Border

The default borders created using specified parameters can be useful for many applications. Still, you may want to create your own borders using sketch tools. Creating a custom border is a common practice at almost every company and school.

To develop a custom border, you may first want to delete any existing borders currently on the sheet. Then, select the **Define New Border** option from the **Format** pull-down menu, or right-click on the **Borders** folder in the **Drawing Resources** folder and select the **Define New Border** pop-up menu option. When you pick the **Define New Border** option, the panel bar automatically enters Drawing Sketch Panel mode and the tools in the **Drawing Sketch Panel** toolbar become active. See Figure 16-67.

Sketch tools allow you to generate your own border design, title block, drawing view, or symbol. Most of the sketch tools available in a drawing file, such as line, circle, and mirror, function exactly the same as those in a model. For information regarding these tools, refer to previous chapters. However, the following sketch tools are available only for use in drawings or contain specific drawing related options:

Adding Text

You have used the **Text** tool in models to create model sketch text. The **Text** tool functions the same in the drawing environment except for the style and property options that become available when adding text to a drawing. The following information describes text preferences specific to adding text to a drawing using the **Format Text** dialog box shown in Figure 16-68:

- **Style.** Use this drop-down list to choose the text style used when adding drawing text. The available styles are defined in the **Styles and Standards Editor**.
- **Property field text.** Property field text is text that is added to a drawing that corresponds to one of the iProperties in the **Properties** dialog box or a prompted text entry. When the border or other drawing element that contains property field text is inserted on a sheet, the property or prompted text created using the **Text** tool is also inserted. The first step in creating property field text is to select the type of field you want to create from the **Type** drop-down list. The following field property types are available:

Figure 16-67.
The panel bar, in Drawing Sketch Panel mode, and the **Drawing Sketch Panel** toolbar contain tools for drawing a custom border.

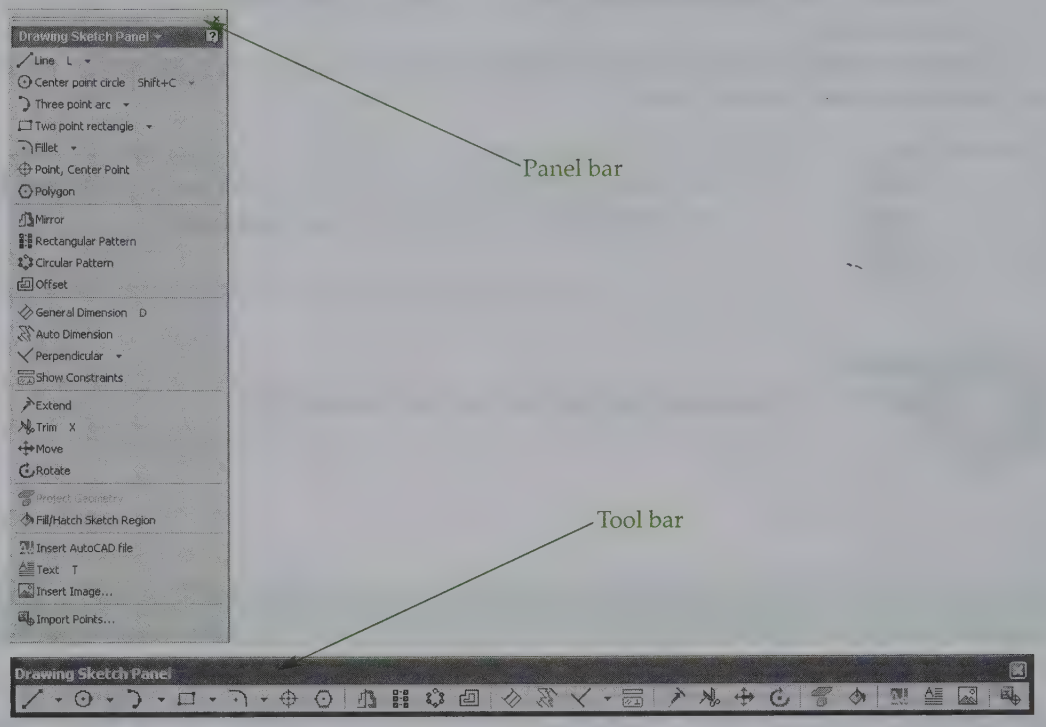
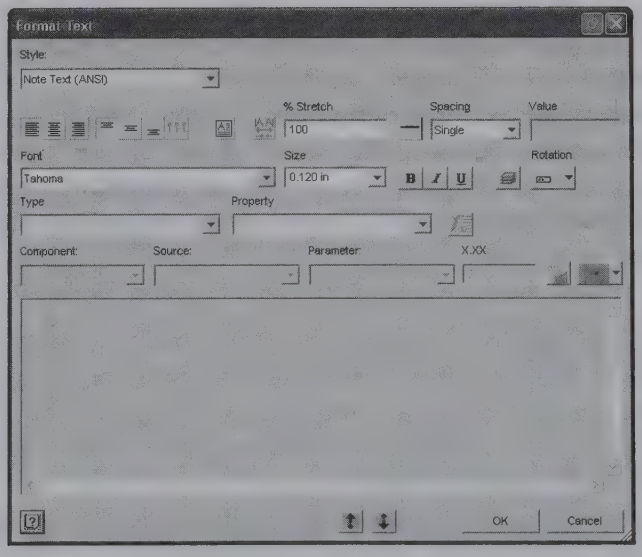


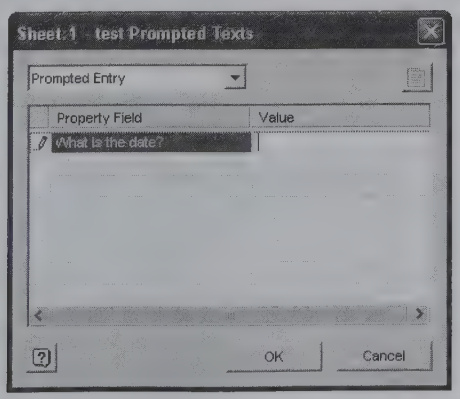
Figure 16-68.
The **Format Text** dialog box.



- **Properties – Model.** These properties correspond to the iProperties you specify in the **Properties** dialog box while developing a model. For example, when you insert a part model view onto the sheet, the properties you specified while in the part file are available to be used in a property field. When this option is selected, the design properties available from the model are listed in the **Property** drop-down list.
- **Properties – Drawing.** These properties correspond to the iProperties you specify in the **Properties** dialog box of the current drawing file. When this option is selected, the design properties available in the drawing are listed in the **Property** drop-down list.
- **Custom Properties.** This option is only available when you create custom properties in the **Properties** dialog box of the current drawing file (Custom Properties – Drawing) or a model inserted in the current drawing as a drawing view (Custom Properties – Model). If a custom property is available and you select the **Custom Properties** option, the custom design properties available in the drawing or model are listed in the **Property** drop-down list.
- **Drawing Properties.** These properties relate to elements of the current drawing, such as the number of sheets. When inserted, the value of the property field reflects the sheet number on which it is inserted. For example, if you insert this property on sheet 1, the value of the property is 1.
- **Sheet Properties.** These properties are similar to drawing properties, because they relate to current drawing elements, such as the sheet number and sheet size. When inserted, the value of the property field reflects the sheet number or sheet size on which it is inserted. For example, if you insert this property on a C-size sheet, the value of the property is C.
- **Prompted Entry.** This option is used to insert a prompted text value on a sheet. Use this technique when you insert the property and want to be prompted for a particular value such as "What is the date?" To create a prompted entry, type the prompt in the **Field Text** edit box. When inserted, the **Edit Property Fields** dialog box is displayed. See Figure 16-69. The **Edit Property Fields** dialog box prompts you with the prompt you specified. Enter the desired value in the edit box and pick the **Check Mark** button to insert the value, or pick the **Delete** button if you do not want to insert a particular prompted value, but do want to insert the border, title block, draft view or symbol.

Once you select the property type and pick the property using the **Property** drop-down list, pick the **Add Text Parameter** button to add the property field to the **Text** edit box. The rest of the options in the **Format Text** dialog box function exactly the same as when adding text to model sketches. For more information regarding the **Text** tool, refer to Chapter 4.

Figure 16-69.
The **Field Entry** dialog box.



Filling and Hatching Sketch Regions

This tool allows you to add a solid color fill or a hatch pattern to a sketch region. See Figure 16-70. A sketch region must be available before you can access the **Hatch/Color Fill** tool, and can consist of any closed-loop piece of geometry, such as a circle or a rectangle. After you create an appropriate sketch region, access the **Hatch/Color Fill** tool by picking the **Fill/Hatch Sketch Region** button on the **Drawing Sketch Panel** bar, or pick the **Fill/Hatch Sketch Region** button on the **Drawing Sketch Panel** toolbar.

Figure 16-70.
Filling or hatching a sketch region.



Once you access the **Hatch/Color Fill** tool, use the cursor to select a sketch region to define the area to fill or hatch. See Figure 16-71. When you select appropriate sketch geometry, the **Hatch/Color Fill** dialog box is displayed. See Figure 16-72. If you want to add hatching to the sketched region, pick the **Enable** check box in the **Hatch** area. Then, pick the hatch pattern you want to use from the **Pattern** drop-down list. Only hatch patterns selected from the **Hatch Style** area of the **Styles and Standards Editor** are available. You can override any of the default properties in the **Hatch** area, and the hatch color. These properties correspond to those previously described in the **Hatch Style** area of the **Styles and Standards Editor**. Changes made to the hatch properties are previewed in the selected sketch region. When you are satisfied with the hatch preview pick the **OK** button to create the hatching.

If you want to add a solid color to the sketched region, pick the **Enable** check box in the **Color Fill** area. Then, specify the color of the fill by selecting the **Color** button and choosing a color from the **Color** dialog box. When you are satisfied with the hatch preview in the selected sketched region, pick the **OK** button to create the fill.

NOTE



In most situations, hatch line weight, and hatch and color fill color should be set to By Layer, which will automatically assign a line weight and color based on the characteristics of the layer defined in the **Object Defaults Style** area for objects with hatch.

Figure 16-71.
Selecting a fill or hatch sketches region.

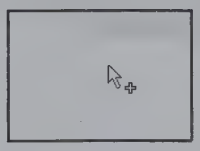
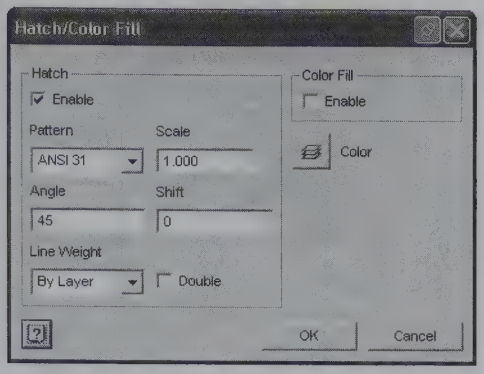


Figure 16-72.
The **Hatch/Color Fill** dialog box.



PROFESSIONAL TIP



Use the **Insert AutoCAD file** tool in the sketch environment to create a border from an existing AutoCAD file.

The final step in creating your own border is to define the location where you want the corner of the title block to be inserted. To specify the insertion point, you must first pick the point you want to use, which is typically the corner of the border. Then, select the **Set insertion Point Grip** option from the **Inventor Standard** toolbar. With a specified insertion point, the title block can be placed in the correct location.

After you sketch a border using sketching tools and define the title block insertion point, you are ready to generate the border. To exit the sketch environment, right-click and select the **Save Border** menu option, or pick the **Return** button on the **Inventor Standard** toolbar, or select the **Save Border** option from the **Format** pull-down menu.

Before you exit the sketch environment, the **Border** dialog box is displayed, which allows you to save the border. See Figure 16-73. Type the name of the border in the **Name** edit box and pick the **Save** button to create the border. If you pick the **Discard** button, the sketch is deleted and the border is not created.

The border you create as a sketch is transformed into an insertable drawing item. Your custom borders, along with any other borders currently in the drawing, are contained in the **Borders** folder of the **Drawing Resources** folder in the browser. See Figure 16-74. Any border contained in the **Borders** folder can be inserted into the current sheet and future sheets by double-clicking the border name or right-clicking the border name and selecting the **Insert** menu option. To edit a custom border, right-click the border name in the **Borders** folder and pick the **Edit** menu option.

Figure 16-73.
The **Border** dialog box.

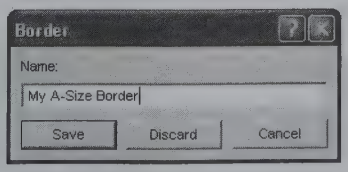
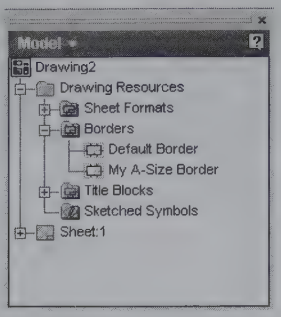
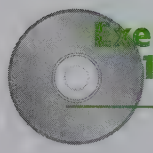


Figure 16-74.
The available drawing borders.



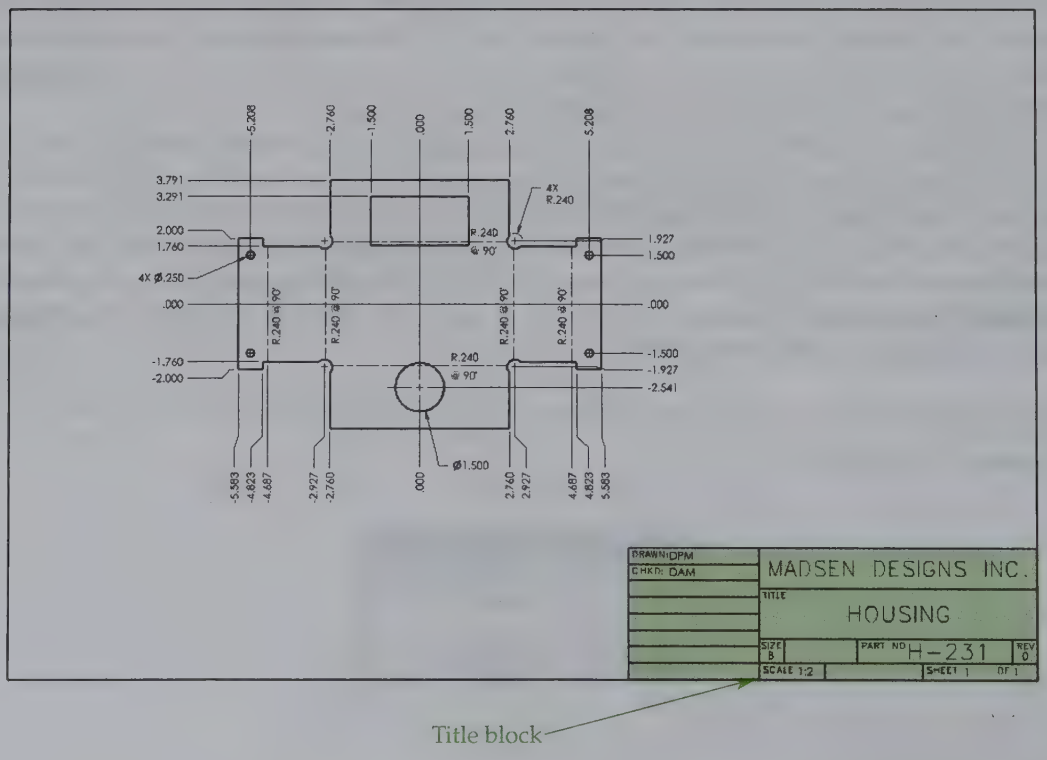


Exercise 16-22 Complete the Exercise on the Student CD.

Using Default Title Blocks

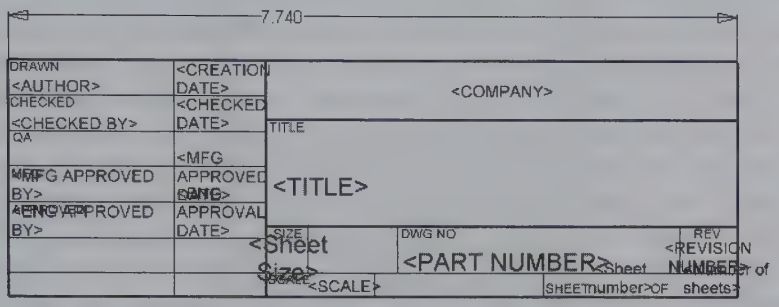
After you insert a border onto the drawing sheet, depending on your approach, the next step in developing a drawing is to define a title block. Title blocks contain information about the model, company, drafter, tolerances, and other design information. A title block can be as simple as a rectangle inside the drawing border with a couple lines of text, or may include several property areas. See Figure 16-75.

Figure 16-75.
An example of a drawing title block.



When you open an Inventor drawing file for the first time, a title block is already placed on the sheet, along with a border. All the text in the default title blocks is created using **Text** tool property fields and corresponding iProperty entries in the **Properties** dialog box. See Figure 16-76. To add information to a default title block, access the **Properties** dialog box from the **File** pull-down menu and enter information in various edit boxes. The information you add is automatically displayed in the specified areas of the title block.

Figure 16-76.
Property fields in a default title block.



Notice in Figure 16-76 that the default large title block contains a scale property field. The scale property field is custom, because there is no scale edit box in the **Properties** dialog box. See Figure 16-77. To enter a scale in the title block, pick the **Custom** tab of the **Properties** dialog box. See Figure 16-78. Then enter the name of the property in the **Name** edit box, or select a name, if it is available, from the **Name** drop-down list. You must enter the name exactly as it was defined when it was created. For example, the scale property is named SCALE. If you enter scale as the name, the value will not be applied to the property field. The next step is to define the type of

Figure 16-77.
The **Properties** dialog box, **Summary** tab shown.

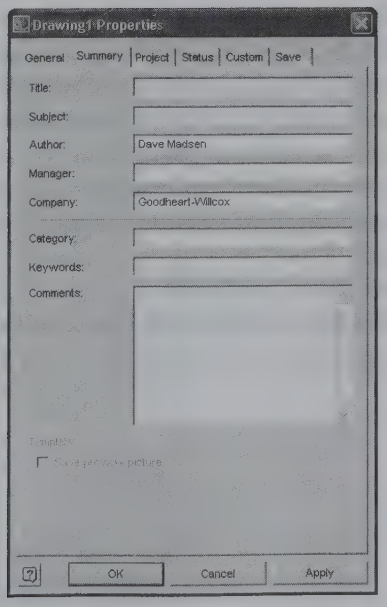
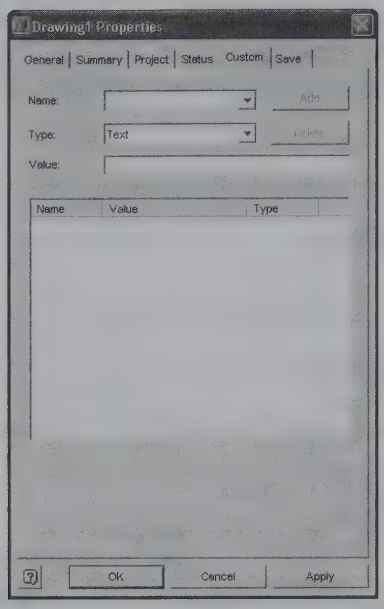


Figure 16-78.
The **Properties** dialog box, **Custom** tab.



property you are creating from the **Type** drop-down list. A property type can be text, a date, numbers, or Yes or No. Then enter a value for the property, such as 1:1 for scale, in the **Value** edit box, and pick the **Add** button to add the custom property to the **Property** list. If you want to delete a custom property, pick the property you want to remove and select the **Delete** button. Finally, pick the **Apply** button to submit the changes.

If you can fully define the default title block using the **Properties** dialog box, you are ready to move on to the next drawing phase. However, if you want to redefine the size, property fields, text, or any other aspect of the current title block on Sheet:1, right-click on the default title block in the **Title Blocks** folder of **Drawing Resources** folder in the browser and select the **Edit** menu option. Use sketching tools to modify the required information and geometry.

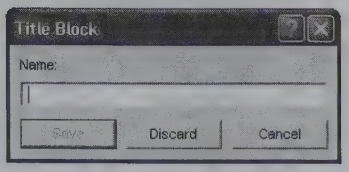
Once you fully define the default title block as needed, exit the sketch environment by right-clicking and selecting the **Save Title Block** menu option, or pick the **Return** button on the **Inventor Standard** toolbar, or select the **Save Title Block** option from the **Format** pull-down menu.

If you redefine a title block sketch you will be prompted to save edits. If you want to save the edits to the original title block, pick the **Save** button. However, if you want to create a copy of the original title block and save the edits to the copy, pick the **Save As...** button to access the **Title Block** dialog box. See Figure 16-79. Type the name of the title block in the **Name** edit box and pick the **Save** button to create the title block. If you pick the **Discard** button, the sketch changes are deleted, and the title block is not modified.



Exercise 16-23 Complete the Exercise on the Student CD.

Figure 16-79.
The **Title Block** dialog box.



Creating a Custom Title Block

The ability to use and modify default title blocks is acceptable for many applications. Still, you may want to create your own title blocks using sketching tools. Creating a custom title block is a common practice at almost every company and school. A custom title block allows you to place a company or school logo or insignia, and design the title block content with your specific information and to suit your needs.

Creating a title block is very similar to editing an existing title block. To develop a custom title block, you may first want to delete any existing title blocks currently on the sheet. Then, select the **Define Title Block** option from the **Format** pull-down menu, or right-click on the **Title Blocks** folder in the **Drawing Recourses** folder and select the **Define New Title Block** pop-up menu option. When you pick the **Define Title Block** option, the panel bar automatically enters **Drawing Sketch Panel** mode and the tools in the **Drawing Sketch Panel** toolbar become active.

The drawing sketch environment is exactly the same when sketching a title block as it is when sketching a border. Refer to the previous discussion on creating your own border for information regarding custom title block development.

PROFESSIONAL TIP

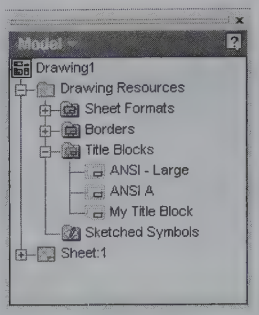


Use the **Insert AutoCAD file** tool in the sketch environment to create a title block from an existing AutoCAD file.

After you sketch a title block using sketching tools, you are ready to generate the title block. To exit the sketch environment, right-click and select the **Save Title Block** menu option, or pick the **Return** button on the **Inventor Standard** toolbar, or select the **Save Title Block** option from the **Format** pull-down menu.

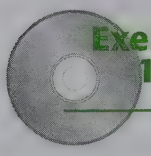
When you exit the sketch environment, the **Title Block** dialog box, previously described, is displayed, which allows you to save the title block using a desired name. The title block you create as a sketch is transformed into an insertable drawing item. Your custom title blocks, along with any other title blocks currently in the drawing, are contained in the **Title Blocks** folder of the **Drawing Resources** folder in the browser. See Figure 16-80. A title block contained in the **Title Blocks** folder can be inserted into the current sheet and future sheets by double-clicking the title block name or right-clicking the title block name and selecting the **Insert** menu option. The insertion point of the title block is defined by the extents of the border and the orientation options in

Figure 16-80.
The available drawing title blocks.

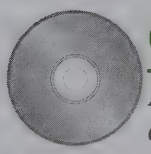


reference to the sheet layout. To adjust the location of the title block, enter the desired orientation in the **Edit Sheet** dialog box. You can also define the location of the title block before insertion by selecting **Application Options** from the **Tools** pull-down menu, and picking the **Drawing** tab.

To edit a custom title block, right-click the title block name in the **Title Blocks** folder and pick the **Edit** menu option.



Exercise 16-24 Complete the Exercise on the Student CD.



Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. List the order used in this textbook to approach the creation of drawings.
2. Describe a *drawing sheet.*
3. What do drawing sheets contain?
4. Briefly explain the function of *drawing styles,* and give at least two examples.
5. List at least three characteristics to remember about drawing styles.
6. Describe *layers* and their use.
7. Give at least two examples of layer use.
8. Discuss how layers are assigned automatically.
9. How do you create a new layer and add the new style to the list of available styles?
10. Identify the characteristics of a Dimension layer; including the layer name, visibility, color, line type, line weight, and scale.
11. Discuss *text styles* and give an example of their use.
12. Why are text styles considered sub-styles?

13. Give a reason for adjusting text style variables rather than using the default values.
14. Give the characteristics of a metric ASME 3 mm high text style, including height, name, and font.
15. Define *balloon*.
16. Define *parts list*.
17. Explain the basic use of *balloon styles*.
18. Explain the use of *center marks* and *centerlines*.
19. Briefly describe the purpose of center mark styles.
20. Describe the basic function of *datum targets* and explain the appearance of their representation, including the leader.
21. Give the basic function of *dimension styles*.
22. Define the term *hatch* or *hatching* and give an example of its use.
23. Briefly discuss the use of *hatch styles*.
24. What is the function of *hole table styles*?
25. Define *datums*.
26. Discuss the use of parts lists and parts list styles.
27. Define *surface finish*, discuss the appearance of the surface finish symbol, and briefly explain the use of the surface texture style.
28. What is a *weldment drawing*?
29. Identify at least two Inventor tools and preferences specific to 2D weldment drawings.
30. What are *object defaults*?
31. Describe a *standard* and its use.
32. What is an *active standard*?
33. Discuss the purpose for updating styles and standards, and which styles and standards can be saved.
34. What do you do if you do *not* want to save title block edits to the original title block?

Problems

Instructions:

- Create an inch drawing template according to the following specifications.
- Use the information provided in this chapter to develop the drawing template.
- When creating styles and standards, change only the settings shown. Do not modify any of the other default style values. This process assumes that the default styles and standards have not been modified and saved to the styles library.
- Once you add a new style or standard, and modify the settings, save the style or standard to the styles library by right-clicking on the new style or standard in the **Styles and Standards Editor** browser, and selecting the **Save to Styles Library** pop-up menu option.
- Save frequently during the development of this template.

1. Create an inch drawing template by saving a copy of the ANSI (in).idw template file, found in Autodesk/Inventor 11/Templates/English/ANSI (in).idw, to your INCH Template folder. Name the copied file DRAWING-in.ipt.
2. Access the **Styles and Standards Editor**.
3. Select the **All Styles** option of the **Filter Style** drop-down list to show all styles.
4. Pick the **Leader** node to display the **Leader Style** area.
5. Create the following leader styles according to the values shown:
   - Arrowhead leader
     - **Existing leader style:** General (ANSI)
     - **New leader style name:** ARROWHEAD-in
     - **Arrowhead:** Filled
     - **Size (X):** .125″
     - **Height (Y):** .042″
     - **Extension Line Offset:** .063″
     - **Extension Line Overshoot:** .125″
     - **Line Type:** By Layer
     - **Line Weight:** By Layer
     - **Color:** By Layer
     - **Comments:** Inch ASME filled arrowhead leader
   - Datum leader
     - **Existing leader style:** ARROWHEAD-in
     - **New leader style name:** DATUM-in
     - **Arrowhead:** Datum 60 Filled
     - **Size (X):** .125″
     - **Extension Line Offset:** .063″
     - **Extension Line Overshoot:** .063″
     - **Line Type:** By Layer
     - **Line Weight:** By Layer
     - **Color:** By Layer
     - **Comments:** Inch ASME datum feature leader
   - Dot leader
     - **Existing leader style:** ARROWHEAD-in
     - **New leader style name:** DOT-in
     - **Arrowhead:** Small Dot
     - **Size (X):** .063″
     - **Extension Line Offset:** .063″
     - **Extension Line Overshoot:** .063″
     - **Line Type:** By Layer
     - **Line Weight:** By Layer
     - **Color:** By Layer
     - **Comments:** Inch ASME dot leader
6. Pick the **Text** node to display the **Text Style** area.
7. Create the following text styles according to the values shown:
   - .10″ high text
     - **Existing text style:** Note Text (ANSI)
     - **New text style name:** .10 in TEXT
     - **Font:** Century Gothic

        - **Text Height:** .10″
        - **Comments:** Inch ASME .10 in high text
    - .12″ high text
        - **Existing text style:** Note Text (ANSI)
        - **New text style name:** .12 in TEXT
        - **Font:** Century Gothic
        - **Text Height:** .12″
        - **Comments:** Inch ASME .12 in high text
    - .24″ High Text
        - **Existing text style:** .12 in TEXT
        - **New text style name:** .24 in TEXT
        - **Font:** Century Gothic
        - **Text Height:** .24″
        - **Comments:** Inch ASME .24 in high text
8. Pick the **Balloon** node to display the **Balloon Style** area.
9. Create a new balloon style named BALLOON-in based on the existing Balloon (ANSI) style, and according to the values shown:
    - **Leader Style:** ARROWHEAD
    - **Alternate Style:** DOT
    - **Text Style:** .24 in TEXT
    - **Default Offset:** .18″
    - **Comments:** Inch ASME balloon.
10. Pick the **Center Mark** node to display the **Center Mark Style** area.
11. Create a new center mark style named CENTER MARK-in based on the existing Center Mark (ANSI) style, and according to the values shown:
    - **Mark:** .125 in
    - **Gap:** .063 in
    - **Overshoot:** .125″
    - **Extension:** .125″
    - **Default Radius:** .5″
    - **Comments:** Inch ASME center mark
12. Pick the **Datum Target** node to display the **Datum Target Style** area.
13. Create a new datum target style named DATUM TARGET-in based on the existing Datum Target (ANSI) style, and according to the values shown:
    - **General** tab:
        - **Leader Style:** ARROWHEAD-in
        - **Text Style:** .12 in TEXT
        - **Target Point Size:** .12″
        - **Comments:** Inch ASME datum target symbol
    - **Units** tab:
        - **Primary Units:** Use Common
        - **Unit String:** Not selected
        - **Leading Zeros:** Not selected
        - **Trailing Zeros:** Selected
        - **Alternate Units:** Not selected
14. Pick the **Dimension** node to display the **Datum Target Style** area.
15. Create a new dimension style named DIMENSION-in based on the existing Default (ANSI) style, and according to the values shown:
    - **Units** tab:
        - **Units Linear:** inch
        - **Linear Precision:** 3.123
        - **Angular Precision:** 3.123

- **Display Units Type:** Not selected
- **Display Leading Zero:** Not selected
- **Display Trailing Zero:** Selected
- **Angular Display Leading Zero:** Not selected
- **Angular Display Trailing Zero:** Not selected
- **Comments:** Inch ASME dimension

- **Alternate Units** tab:
  - **Display Alternate Units:** Not selected
- **Display** tab:
  - **Terminator Size (X):** .125″
  - **Terminator Height (Y):** .042″
  - **Extension:** .125″
  - **Origin Offset:** .063″
  - **Gap:** .063″
  - **Spacing:** .24″
  - **Part Offset:** .375″
- **Text** tab:
  - **Primary Text Style:** .12 in TEXT
  - **Size:** .12″
- **Tolerance** tab:
  - **Angular Precision:** 3.123
  - **Leading Zero:** Not selected
  - **Trailing Zero:** Selected
  - **Zero Tolerance Display:** Full Display
  - **Zero Arc Min/Sec Display:** Hide Arc Min/Sec
- **Options** tab:
  - **Hide Origin Indicator:** Selected
- **Notes and Leaders** tab:
  - **Hole Note Settings:** Selected
    - **Use Default:** Selected
  - **Chamfer Note Settings:** Selected
    Replace <ANGL> with <DIST1>
    Replace <DIST1> with <ANGL>
    Delete the word "Chamfer"
    - **Leader Style:** ARROWHEAD-in

16. Create a new dimension style named ORDINATE-in based on the existing DIMENSION-in style, and according to the values shown:
    - **Text** tab:
      - **Vertical Dimension:** Inline - Vertical
17. Pick the **Feature Control Frame** node to display the **Feature Control Frame Style** area.
18. Create a new feature control frame style named FEATURE CONTROL FRAME-in based on the existing Feature Control Frame (ANSI) style, and according to the values shown:
    - **General** tab:
      - **Allow Tolerance 2:** Not selected
      - **Leader Style:** ARROWHEAD-in
      - **Text Style:** .12 in TEXT
      - **White Space:** .063″
      - **Comments:** Inch ASME feature control frame

- **Units** tab:
  - **Primary Units:** Use Common
  - **Unit String:** Not selected
  - **Leading Zeros:** Not selected
  - **Trailing Zeros:** Selected
  - **Alternate Units:** Not selected

19. Pick the **Hole Table** node to display the **Hole Table Style** area.
20. Create a new hole table style named HOLE TABLE-in based on the existing Hole Table (ANSI) style, and according to the values shown:
    - **Formatting** tab:
      - **Title:** HOLE TABLE
      - **Text Style:** .12 in TEXT
      - Right-click on the XDIM property and pick the **Format Column** menu option. Then, deselect the **Leading Zero** check box in the **Display** area.
      - Right-click on the YDIM property and pick the **Format Column** menu option. Then, deselect the **Leading Zero** check box in the **Display** area.
      - **Comments:** Inch ASME hole table
21. Pick the **ID** node to display the **ID Style** area.
22. Create a new ID style named DATUM FEATURE-in based on the existing Datum ID (ANSI) style, and according to the values shown:
    - **Leader Style:** DATUM-in
    - **Text Style:** .12 in TEXT
    - **Orientation:** Always Horizontal
    - **Comments:** Inch ASME datum feature symbol
23. Pick the **Parts List** node to display the **Parts List Style** area.
24. Create a new parts list style named PARTS LIST-in based on the existing Parts List (ANSI) style, and according to the values shown:
    - **Title Name:** PARTS LIST
    - **Text Style:** .12 in TEXT
    - **Heading Gap:** .063″
    - **Row Gap:** .063″
    - **Direction:** Add new parts to top
    - **Heading:** Bottom
    - ITEM Column width: 1″
    - QTY Column width: 1″
    - PART NUMBER Column width: 2.125″
    - DESCRIPTION Column width: 2.125″
    - **Comments:** Inch ASME parts list
25. Pick the **Surface Texture** node to display the **Surface Texture Style** area.
26. Create a new surface texture style named SURFACE TEXTURE-in based on the existing Surface Texture (ANSI) style, and according to the values shown:
    - **Leader Style:** ARROWHEAD-in
    - **Text Style:** .12 in TEXT
    - **Symbol Size:** Small (pick button)
    - **Comments:** Inch ASME surface texture symbol
27. Pick the **Table** node to display the **Table Style** area.

28. Create a new table style named TABLE-in based on the existing Table (ANSI) style and according to the values shown:
    - **Title Name:** TABLE
    - **Title Text Style:** .24 in TEXT
    - **Title Text Style:** .12 in TEXT
    - **Title Text Style:** .12 in TEXT
    - **Heading Gap:** .063″
    - **Row Gap:** .063″
    - **Direction:** Add new parts to bottom
    - **Heading:** Top
    - **Comments:** Inch ASME table.
29. Pick the **Weld Symbol** node to display the **Weld Symbol Style** area.
30. Create a new weld symbol style named WELD SYMBOL-in based on the existing Weld Symbol (ANSI) style, and according to the values shown:
    - **Leader Style:** ARROWHEAD-in
    - **Text Style:** .12 mm TEXT
    - **Comments:** Inch AWS/ASME weld symbol
31. Pick the **Weld Bead** node to display the **Weld Bead Style** area.
32. Create a new weld bead style named WELD BEAD-in based on the existing Weld Bead Recovery (ANSI) style, and according to the values shown:
    - **Comments:** Inch AWS/ASME weld bead
33. Pick the **Object Defaults** node to display the **Object Defaults Style** area.
34. Create a new object defaults style named OBJECT DEFAULTS-in based on the existing Object Defaults (ANSI) style, and according to the values shown:
    - **Filter:** All Objects
    - **Comments:** Inch ASME object defaults
    - **3D Sketch Geometry:** Layer: 3D Sketch Geometry (ANSI)
    - **Angle Dimension: Object Style:** DIMENSION-in, **Layer:** DIMENSION
    - **Balloon: Object Style:** BALLOON-in, **Layer:** ASSEMBLY ANNOTATION
    - **Baseline Dimension: Object Style:** DIMENSION-in, **Layer:** DIMENSION
    - **Border Geometry: Layer:** SHEET
    - **Border Text: Object Style:** .12 in TEXT, **Layer:** SHEET TEXT
    - **Broken View Line: Layer:** BREAK
    - **Caterpillar: Object Style:** WELD BEAD-in, **Layer:** WELDMENT
    - **Center Mark: Object Style:** CENTER MARK-in, **Layer:** CENTERLINE
    - **Centerline: Object Style:** CENTER MARK-in, **Layer:** CENTERLINE
    - **Datum Id: Object Style:** DATUM FEATURE-in, **Layer:** DIMENSION
    - **Datum Target: Object Style:** DATUM TARGET-in, **Layer:** DIMENSION
    - **Detail Cut Line: Layer:** DETAIL BREAK
    - **Detail View Callout: Object Style:** DIMENSION-in, **Layer:** DETAIL CALLOUT
    - **Diameter Dimension: Object Style:** DIMENSION-in, **Layer:** DIMENSION
    - **End Treatment: Object Style:** WELD BEAD-in, **Layer:** WELDMENT
    - **Feature Control Frame: Object Style:** FEATURE CONTROL FRAME-in, **Layer:** DIMENSION
    - **Feature Id: Object Style:** DATUM FEATURE-in, **Layer:** DIMENSION
    - **Hatching: Object Style:** HATCH, **Layer:** HATCH
    - **Hidden Tangent Edge: Layer:** HIDDEN
    - **Hidden Thread Ends: Layer:** HIDDEN
    - **Hidden Thread Lines: Layer:** HIDDEN
    - **Hidden View Edge: Layer:** HIDDEN
    - **Hole Note: Object Style:** DIMENSION-in, **Layer:** DIMENSION
    - **Hole Table: Object Style:** HOLE TABLE-in, **Layer:** DIMENSION
    - **Hole Tag: Object Style:** DIMENSION-in, **Layer:** DIMENSION

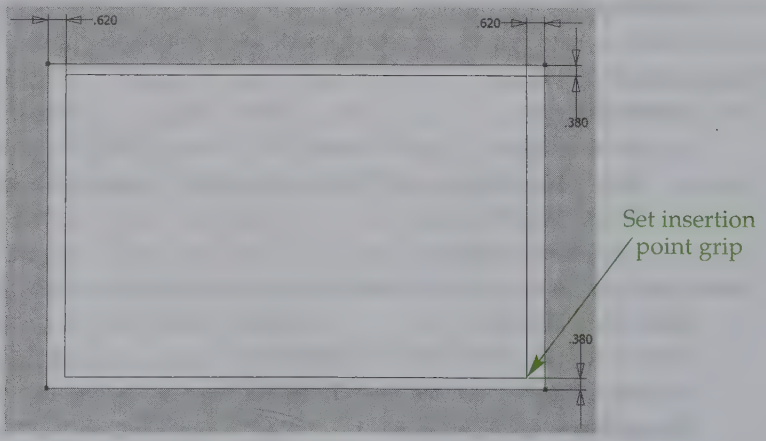
- **Leader Text: Object Style:** DIMENSION-in, **Layer:** DIMENSION
- **Linear Dimension: Object Style:** DIMENSION-in, **Layer:** DIMENSION
- **Ordinate Dimension: Object Style:** ORDINATE-in, **Layer:** DIMENSION
- **Ordinate Dimension Set: Object Style:** ORDINATE-in, **Layer:** DIMENSION
- **Origin Indicator: Object Style:** ARROWHEAD-in, **Layer:** DIMENSION
- **Overlay View: Layer:** ALTERNATE POSITION
- **Parts List: Object Style:** PARTS LIST-in, **Layer:** ASSEMBLY ANNOTATION
- **Presentation View Trail Line: Layer:** ASSEMBLY TRAIL
- **Radial Dimension: Object Style:** DIMENSION-in, **Layer:** DIMENSION
- **Reference Part Edge: Layer:** CHAIN
- **Revision Table: Object Style:** .12 in TEXT, **Layer:** REVISION
- **Revision Tag: Object Style:** DIMENSION-in, **Layer:** REVISION
- **Section View Line: Object Style:** DIMENSION-in, **Layer:** SECTION
- **Sheet Metal Bend Centerline: Layer:** BEND CENTERLINE
- **Sheet Metal Bend Extent: Layer:** BEND
- **Sketch Fill Hatch: Object Style:** HATCH, **Layer:** HATCH
- **Sketch Geometry: Layer:** SKETCH
- **Sketch Text: Object Style:** .12 in TEXT, **Layer:** SKETCH TEXT
- **Surface Texture: Object Style:** SURFACE TEXTURE-in, **Layer:** DIMENSION
- **Text (General Note): Object Style:** .12 in TEXT, **Layer:** NOTE
- **Thread Ends: Layer:** VISIBLE OBJECT
- **Thread Lines: Layer:** HIDDEN
- **Thread Note: Object Style:** DIMENSION-in, **Layer:** DIMENSION
- **Title Block Geometry: Layer:** SHEET
- **Title Block Text: Object Style:** .12 in TEXT, **Layer:** SHEET TEXT
- **User Defined Symbol (Leader Only): Object Style:** ARROWHEAD-in, **Layer:** DIMENSION
- **User Defined Symbol Geometry: Layer:** SKETCH
- **User Defined Symbol Text: Object Style:** .12 in TEXT, **Layer:** SKETCH TEXT
- **View / Scale Label: Object Style:** .12 in TEXT, **Layer:** LABEL
- **Visible Tangent Edge: Layer:** TANGENT EDGE
- **Visible View Edge: Layer:** VISIBLE EDGE
- **Weld Symbol: Object Style:** WELD SYMBOL-in, **Layer:** DIMENSION
- **Work Axis: Object Style:** CENTER MARK-in, **Layer:** WORK AXIS
- **Work Plane: Object Style:** CENTER MARK-in, **Layer:** WORK PLANE
- **Work Point: Object Style:** CENTER MARK-in, **Layer:** WORK POINT

35. Pick the **Standards** node to display the **Standard** area.
36. Create a new standard named ASME – INCH based on the existing Default Standard (ANSI) style, and according to the values shown:
    - **General** tab:
        - **Preset Values:**
            - **Line Weight:** Create a new .012 in and .024 in line weight. Delete all line weights except for the new .012 in and .024 in line weights.
            - **Text Height:** Delete all text heights except for the .1 in, .12 in, and .24 in text heights.
    - **Available Styles** tab: Select the following styles only. Deselect all other styles:
        - **Balloon:** BALLOON-in
        - **Center Mark:** CENTER MARK-in
        - **Datum Target:** DATUM TARGET-in
        - **Dimension:** DIMENSION-in
        - **Feature Control Frame:** FEATURE CONTROL FRAME-in
        - **Hatch:** HATCH
        - **Hole Table:** HOLE TABLE-in
        - **ID:** DATUM FEATURE-in

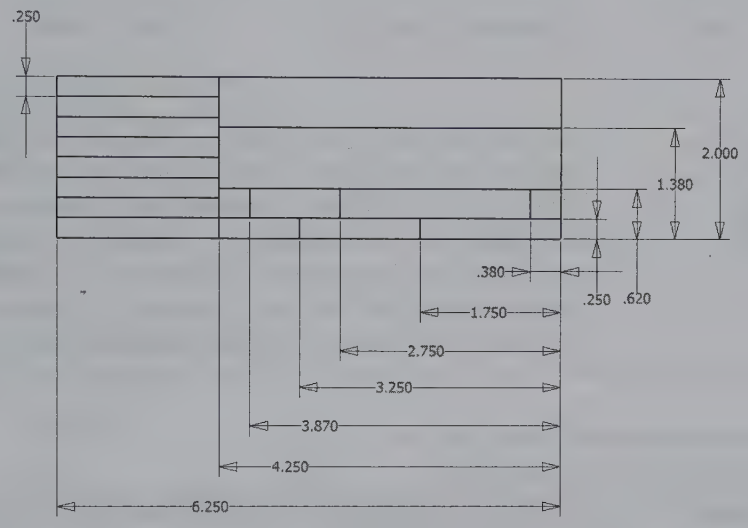
- **Layers:** ALTERNATE POSITION, ASSEMBLY ANNOTATION, ASSEMBLY TRAIL, BEND CENTERLINE, BEND, BREAK, CENTERLINE, CHAIN, DETAIL BREAK, DETAIL CALLOUT, DIMENSION, HATCH, HIDDEN, LABEL, NOTE, SECTION, SHEET, SHEET TEXT, SKETCH, SKETCH TEXT, WELDMENT, REVISION, TANGENT EDGE, VISIBLE OBJECT, WORK AXIS, WORK PLANE, WORK POINT
- **Leader:** ARROWHEAD-mm, DATUM-mm, DOT-in
- **Parts List:** PARTS LIST-in
- **Surface Texture:** SURFACE TEXTURE-in
- **Text:** 3 mm TEXT, 6 mm TEXT
- **Weld Bead:** WELD BEAD-in
- **Weld Symbol:** WELD SYMBOL-in

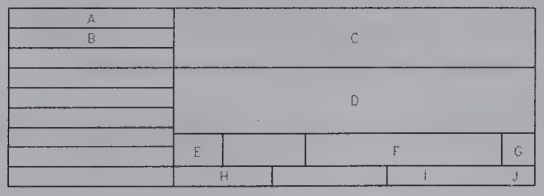
- **Object Defaults tab: Active Object Defaults:** OBJECT DEFAULTS-in

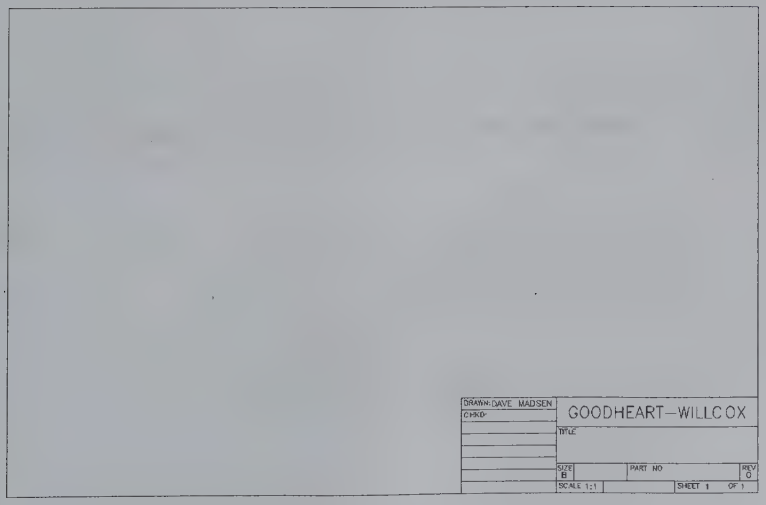
37. Pick the **Save** button to save the changes to the current standard. Then pick the **Done** button to close the **Styles and Standards Editor**.
38. Save all styles and standards to the styles library by selecting the **Save Styles To Styles Library** option of the **Format** pull-down menu to access the **Save Styles To Styles Library** dialog box. Save all the styles to the styles library by picking the **Yes to All** button, followed by picking **OK**, and the **YES** button when you see the alert box.
39. Access the **Document Settings** dialog box, and adjust the following settings:
    - Select the **ASME - INCH** standard from the **Active Standard** drop-down list of the **Standard** tab.
    - Pick the **Drawing** tab and select **Percentage of Dimension Line** from the **Dimension Text Alignment** drop-down list.
    - Choose the **Sheet** tab and change the label in the **Sheet** edit box to SHEET.
40. Apply changes and close the **Document Settings** dialog box.
41. Access the **Options** dialog box, and adjust the following setting in the **Drawing** tab:
    - Ensure the **Display Line Weights** check box is selected, and pick the **Display Line Weights True** radio button.
42. Apply changes and close the **Options** dialog box.
43. Right-click on Sheet:1 in the browser and select the **Edit Sheet...** menu option.
44. Change the sheet size to B, but leave all other settings according to default.
45. Delete the default border and title block in Sheet:1.
46. Access the **Define New Border** tool.
47. Sketch and dimension the rectangle shown.



48. Pick the point shown and change its style to **Set insertion Point Grip.**
49. Save the border as BORDER-in.
50. Insert BORDER on the sheet.
51. Open the **Properties** dialog box, and enter your name in the **Author** edit box, your school or company name in the **Company** edit box, a value of 0 in the **Revision Number** edit box, your instructor's or supervisor's name in the **Checked By** edit box.
52. Specify the following options in the **Custom** tab:
    - Name: SCALE
    - Type: Text
    - Value: 1:1
53. Pick the **Add** button to add the custom property to the **Custom Property** list box.
54. Apply the settings and close the **Properties** dialog box.
55. Access the **Define New Title Block** tool.
56. Use sketch tools to create the title block geometry shown.







57. Add the text and property fields shown by referencing the information in the table below with the letters shown. Dimension and constrain the text as needed.

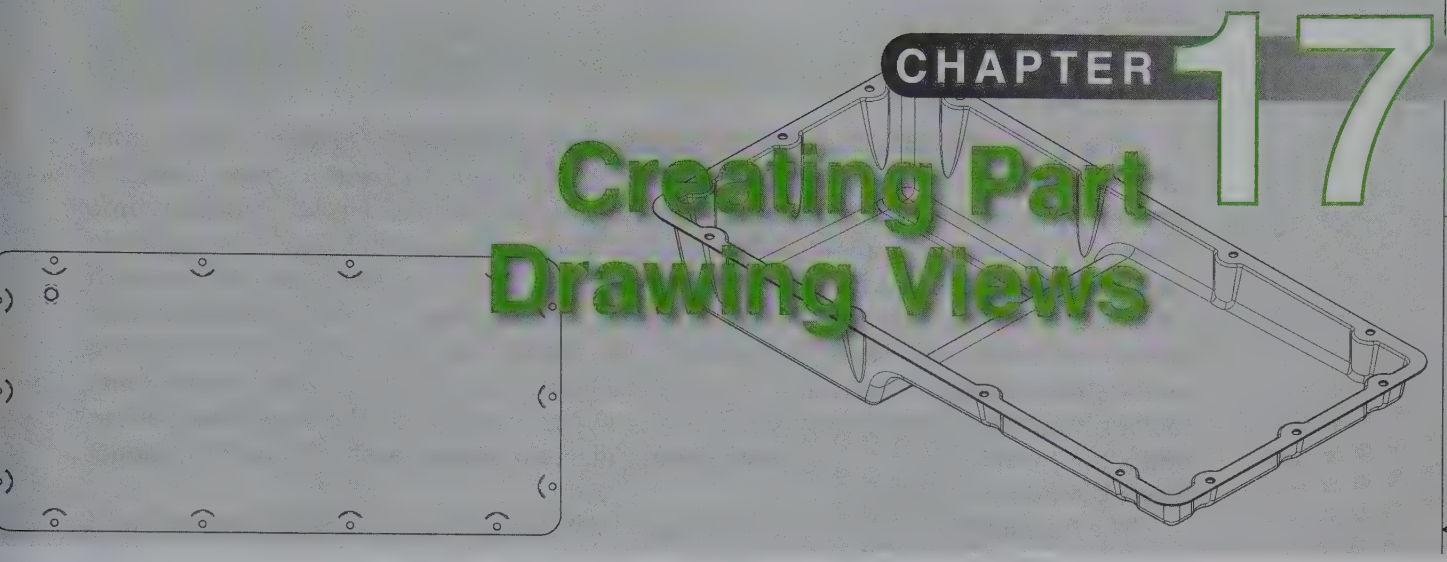
Key	Block Heading	Property Field Text	
A	DRAWN: (.10 in text)	Properties – Drawing	AUTHOR (.12 in TEXT)
B	CHKD: (.10 in text)	Properties – Drawing	CHECKED BY (.12 in TEXT)
C	None	Properties – Drawing	COMPANY (.24 in text)
D	TITLE (.10 in text)	Properties – Model	TITLE (.24 in text)
E	SIZE (.10 in text)	Sheet Properties	Sheet Size (.12 in TEXT)
F	PART NO (.10 in text)	Properties – Model	PART NUMBER (.24 in text)
G	REV (.10 in text)	Properties – Drawing	REVISION NUMBER (.12 in TEXT)
H	SCALE (.10 in text)	Custom Properties – Drawing	SCALE (.12 in TEXT)
I	SHEET (.10 in text)	Sheet Properties – Drawing	Sheet number (.12 in TEXT)
J	OF (.10 in text)	Drawing Properties	Number of sheets (.12 in TEXT)

NOTE



Text added to a title block is defined according to the specified object default, even if you select an alternative text style. For example, even if you pick the .24 in TEXT style, text will be generated using the .12 in TEXT style. In order to produce the correct text height, you must override the text height value in the **Format Text** dialog box.

58. Save the title block as TITLE BLOCK-in.
59. Insert TITLE BLOCK onto the sheet.
60. If you changed the corner point style to **Set insertion Point Grip** when you created the border and have specified the bottom-right-corner title block orientation, the title block should automatically be inserted correctly.
61. The final drawing should look like the drawing shown.
62. Resave DRAWING-in.idw.
63. Exit Inventor without saving, or continue working with the program.



CHAPTER 17

Creating Part Drawing Views

Learning Objectives

After completing this chapter, you will be able to do the following:

- Create and edit base views.
- Project and modify multiviews.
- Create and edit auxiliary views.
- Develop section and broken-out section views.
- Create and modify detail views.
- Create and edit conventional breaks.
- Sketch drawing views.
- Use sheet formats.

Inventor contains tools and options that can be used to develop fully detailed part drawings by referencing existing part models. You can prepare a variety of 2D drawings, including monodetail and multidetail documents. A ***monodetail*** drawing is a drawing of a single part on one sheet, and a ***multidetail*** drawing is a drawing of several parts on one sheet. Once you insert a part model view, you can create additional views, including projected multiviews, isometric, auxiliary, section, detail, conventional break, perspective, and broken out views. In addition, a variety of dimension, note, and text tools are available to fully define drawing geometry. Though 2D drawings are most effectively built by referencing existing models, you can produce a 2D drawing using sketch tools.

Drawing view tools are used to create drawing views of existing part models, new sheets, and draft views. Drawing view tools are accessed from the **Drawing Views Panel** toolbar, the **Drawing Views Panel** bar, or the **Model Views** cascading submenu of the **Insert** pull-down menu.

NOTE



This chapter explores the creation of part drawing views, though all the tools and options described in this chapter are required for creating additional part drawings and assembly drawings.

Creating Base Views

Often, the first view added to a drawing sheet is the base view, such as the front view shown in Figure 17-1. The **Base View** tool allows you to insert a single view, referenced by an existing model, into the drawing. Though you can use this tool anytime during drawing development and generate a wide variety of model views, typically the **Base View** tool is used to place the initial drawing view, or ***base view***, from which projected, isometric, auxiliary, section, detail, conventional break, perspective, and broken out views are added. To access the **Base View** tool, pick the **Base View** button on the **Drawing Views Panel** bar; pick the **Base View** button on the **Drawing Views Panel** toolbar; right-click on a sheet in the graphics window and select the **Base View...** menu option; or from the **Insert** pull-down menu, select the **Base View...** option of the **Model Views** cascading submenu.

The **Drawing View** dialog box is displayed when you access the **Base View** tool, and contains three tabs that allow you to select, define, and place a drawing base view. The **Component** tab, shown in Figure 17-2, is used to select the model file, which is the first step in creating a base drawing view. If model files are currently open, the open files are displayed in the **File** drop-down list and can be selected for use as a base view. If model files are not open, or if you want to reference a model that is not open, you must pick the **Browse** button to access the **Open** dialog box. Use the **Open** dialog box to locate an existing model file, and then pick the **Open** button. The file and its directory should be displayed in the **File** drop-down list. The content of the **Component** tab varies depending on the model you select to use for the base view. When you pick a standard part model, only the **File** drop-down list and **Browse** button is shown. When you select a sheet metal part, the **Sheet Metal View** area is shown. See Figure 17-3. Pick the **Folded Model** radio button to create a base view of the folded sheet metal part, or select the **Flat Pattern** radio button to make a base view of the flat sheet metal pattern. Additional **Component** tab options are provided when creating assembly, weldment, and presentation drawing views. These options are described in Chapter 19.

Figure 17-1.
An initial drawing view, a front view.

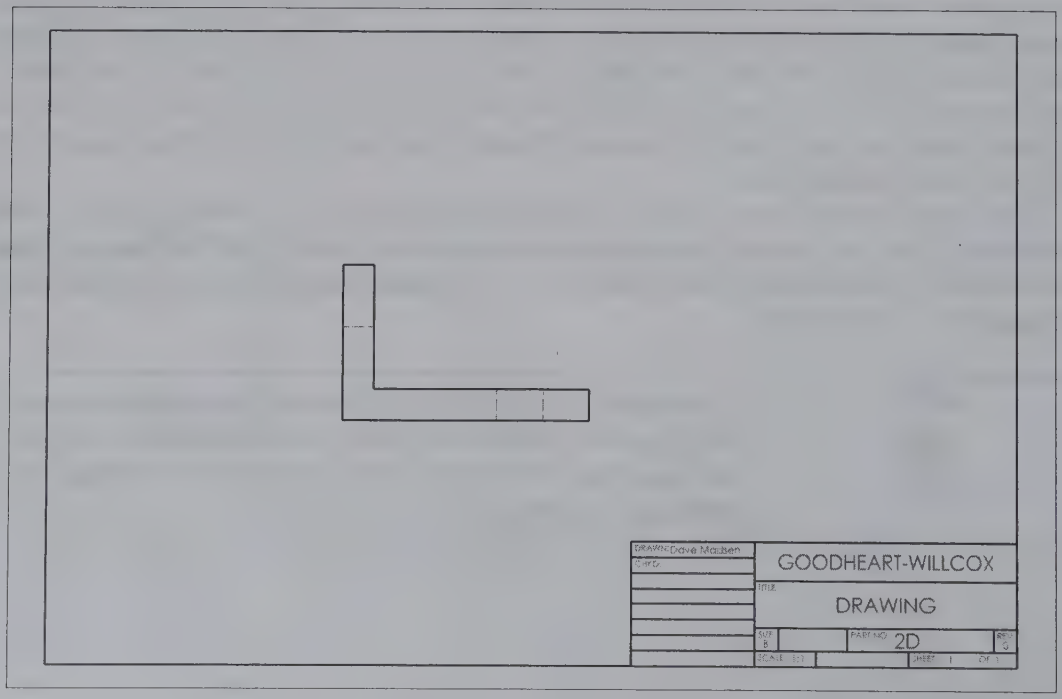


Figure 17-2.
The **Drawing View** dialog box, **Component** tab.

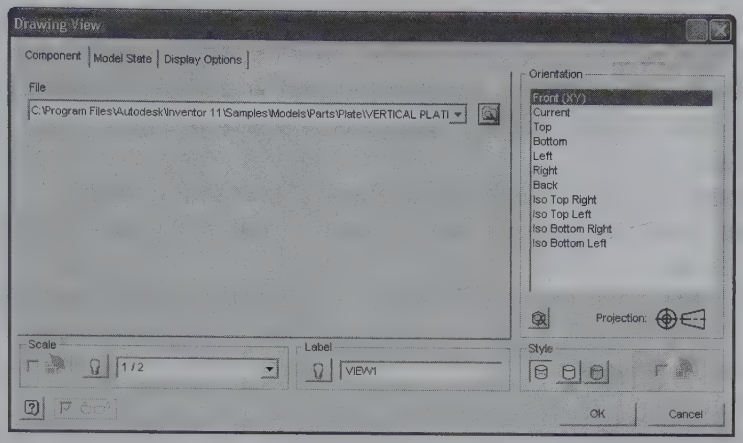
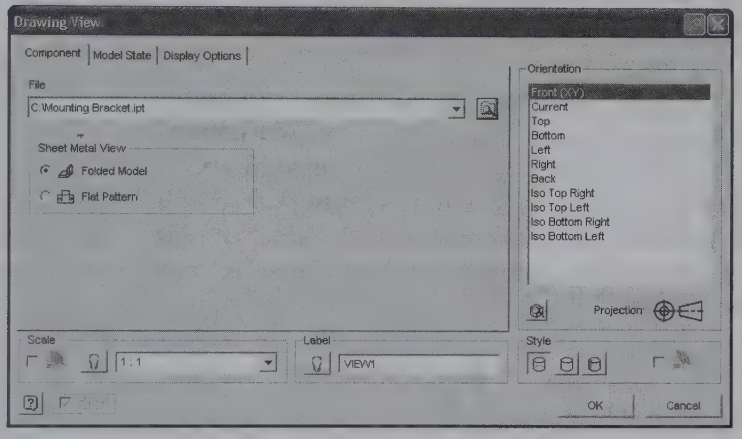


Figure 17-3.
The **Sheet Metal View** area of the **Drawing View** dialog box, **Component** tab.



The **Model State** tab contains options specific to defining assembly, weldment, iPart, and iAssembly drawing base views. The **Model State** tab is described in Chapter 19. The **Display Options** tab, shown in **Figure 17-10,** contains check boxes and drop-down lists that correspond to items that can be displayed in the drawing view, depending on the type of file you selected in the **Component** tab. For example, if you are placing a part file, only part display option check boxes are available. When you select a check box, the specified item is shown in the drawing view. The following information briefly describes each display option:

- **All Model Dimensions.** Select this check box to display the dimensions used to create the model. Only model dimensions that are planar to the specified view are shown.
- **Model Welding Symbols.** Pick this check box to display weld symbols applied to a weldment model.
- **Bend Extents.** Select this check box to show the extents of sheet metal bends.
- **Thread Feature.** Pick this check box to display model thread features.
- **Weld Annotations.** Pick this check box to display weld annotations, such as caterpillars and end treatments, applied to a weldment.

- **Work Features.** Select this check box if you want to show work features placed in the selected model. Only work features that are normal to the specified view orientation are displayed.
- **Tangent Edges.** Select this check box to display tangent model edges. If the **Tangent Edges** check box is selected, the **Foreshortened** check box becomes available allowing you to shorten tangent edges in an effort to differentiate tangent edges from visible object lines, assuming you do not use a separate tangent edge layer.
- **Show Trails.** Pick this check box to show presentation trails.
- **Hatching.** Select this check box to show a hatch pattern applied to a model section view.
- **Align to Base.** Pick this check box to align additional, dependent views with the base view.
- **Definition in Base View.** Pick this check box if you want to display the view projection line.
- **Section Standard Parts.** This drop-down list allows you to define the sectioning preference of standard parts in an assembly drawing view, and functions the same as the **Section Standard Parts** drop-down list found in the **Drawing** tab of the **Options** dialog box.
- **View Justification.** This drop-down list is used to set the default position of drawing views, and functions the same as the **View Justification** drop-down list found in the **Drawing** tab of the **Options** dialog box.

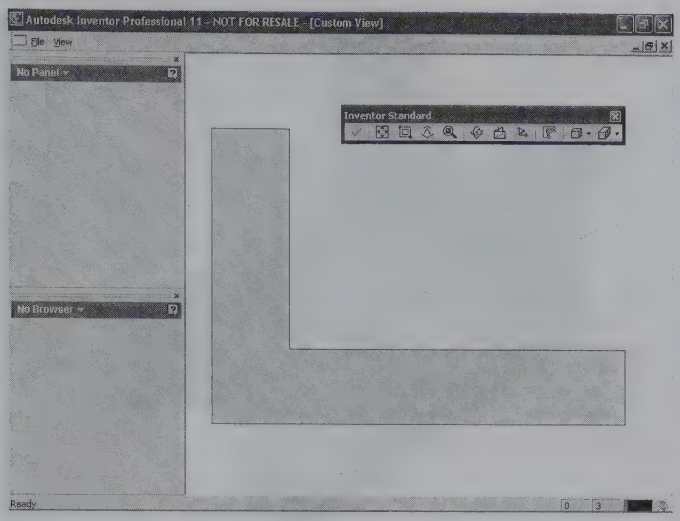
Once you define all the base view component preferences in the **Component**, **Model State**, and **Display Options** tabs, additional orientation and display options are available to further control the base view. The **Orientation** area allows you to define the drawing view used as the base view. The selection box contains a number of possible views. When you pick a view, such as **Front (XY)** or **Top**, the preview of the drawing view in the graphics window changes to reflect the view selection and the X, Y, Z orientation of the selected component. You can select any of the views available to create the drawing base view, but for most part drawings, the initial view should be front, top, or side, because these views allow you to project additional views, including isometrics. For a typically multiview drawing, the base view should be the front view. Multiview drafting theory identifies how the front view should provide the best shape description and most characteristic contours, have the longest dimension, have the fewest hidden features, and is the most stable and natural position.

You should develop models according to the drawing they will be used to create. This means that when you pick the **Front(XY)** orientation option, for example, the correct front view of the model is produced in the drawing. However, creating a model that corresponds to specific drawing views can be difficult, especially for complex or significantly redesigned models. As a result, you may have to select an alternative base view orientation in order to achieve the correct drawing view. Using the previous example, you may pick the **Right** or **Top** orientation option when creating a front view. The alternative is to redefine model sketches or recreate model geometry, which can be tedious and time consuming.

In addition to the standard views available in the **Orientation** area, you can also create a custom view by picking the **Custom View** button. Selecting the **Custom View** button opens the **Custom View** window of Inventor, shown in Figure 17-4. The **Custom View** window allows you to develop a custom drawing view for applications when none of the standard views available in the **Orientation** area is appropriate. The **Custom View** window also allows you to create a perspective view. The **Custom View** window contains many of the typical Inventor viewing tools, available from the **View** pull-down menu and **Inventor Standard** toolbar.

The **Zoom All**, **Zoom Window**, **Zoom**, **Zoom Select**, and **Pan** tools are used for display purposes only, while the **Rotate**, **Look At**, **Rotate at Angle**, **Orthographic Camera**,

Figure 17-4.
The **Custom View** window.



and **Perspective Camera** tools are used to actually define the custom view. For example, if you rotate the front view of an object 10°, a 10° custom view, as shown in Figure 17-5 is created. Another example is creating a perspective view, as shown in Figure 17-6. A *perspective drawing view* is sometimes used to display drawing model geometry in a perspective appearance. In contrast to the more traditional isometric view, a perspective view looks more realistic. This is because depth and distortion are added in an effort to illustrate what the human eye would actually see if the model was a real object. Perspective views are created by rotating the model, and selecting the **Perspective Camera** tool.

The only tool available in the **Custom View** window with which you may not be familiar is the **Rotate at Angle** tool. The **Rotate at Angle** tool allows you to rotate a view at a specified angle. This tool is accessed by picking the **Rotate at Angle** button on the **Inventor Standard** toolbar, or selecting the **Rotate at Angle** option from the **Rotate** cascading submenu of the **View** pull-down menu. When you access the **Rotate at Angle** tool, the **Incremental View Rotate** dialog box is displayed. See Figure 17-7. To use the **Incremental View Rotate** dialog box, enter the desired amount of rotation in the **Increment** edit box. Then, use the rotate buttons to rotate the view to the desired location. Each time you select one of the rotate buttons, the view rotates in the specified direction according to the defined increment. If you want to return to the original view, pick the **Reset** button. Once you rotate the view as desired, pick the **OK** button to exit the **Rotate at Angle** tool. When you are satisfied with the custom view display, exit the

Figure 17-5.
An example of a custom drawing view, a 10° custom view.



Front view 10° rotation custom view

Figure 17-6.
Creating a perspective view using rotate tools and the **Perspective Camera** tool.

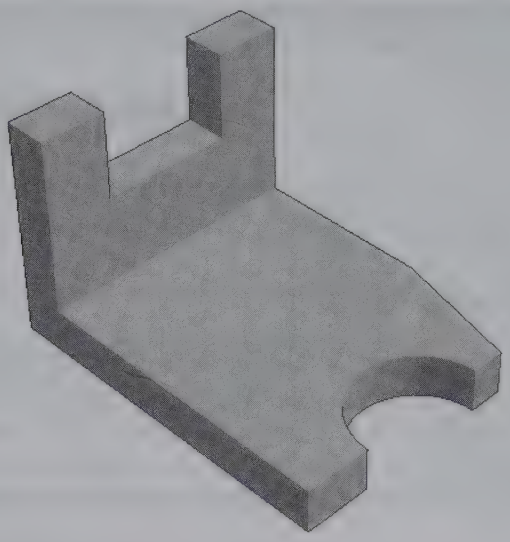
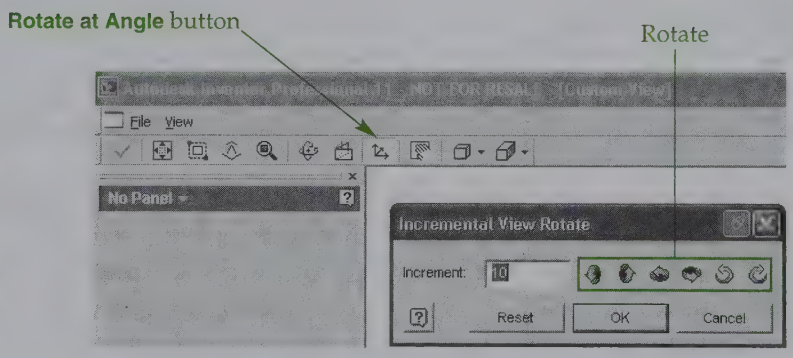


Figure 17-7.
The **Incremental View Rotate** dialog box.



Custom View window by picking the **Exit Custom View** button on the **Inventor Standard** toolbar, or selecting the **Exit Custom** option from the **File** pull-down menu.

The **Scale** area allows you to specify the base view scale and scale display options in relation to the actual model size. Enter the required scale in the **Scale** edit box, or select a scale from the flyout button. The **Scale from Base** check box is not available when you insert the first base drawing view, because the base view is not dependent on another view. However, the **Toggle Scale Visibility** button is offered, which, if selected, displays the view scale under the drawing view on the sheet. See **Figure 17-8.**

The **Label** area allows you to define the base drawing view label. See **Figure 17-9.** You can edit the default name of the label in the **Label** edit box, and pick the **Toggle Label Visibility** button if you want the label displayed in the view. See **Figure 17-10.**

Use the **Style** area to specify the display style of view geometry. You can choose to display the view with hidden lines by picking the **Hidden Line** button, without hidden lines by picking the **Hidden Line Removed** button, or as shaded by picking the **Shaded** button. The **Style from Base** check box is not available when you insert the first base drawing view, because the base view is not a dependent of another view.

Figure 17-8.
An example of a visible scale label.

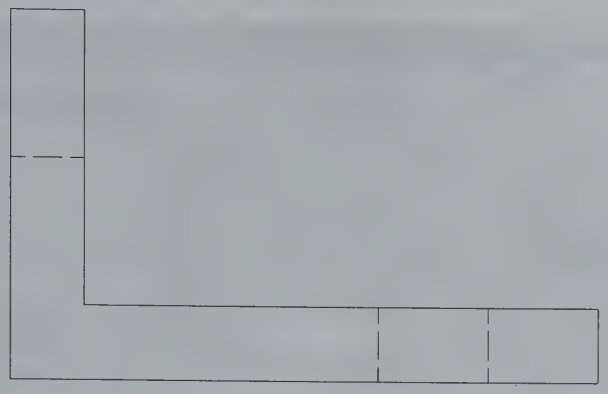
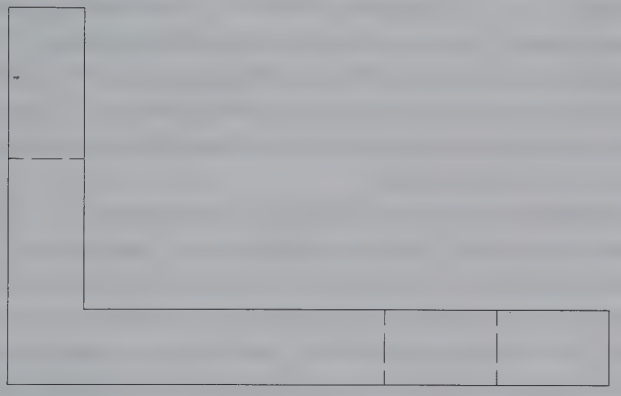


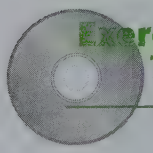
Figure 17-9.
An example of a view label.



Once you select a component to place on the drawing sheet, specify the desired view and define all other view options, pick the location on the sheet where you want to place the base view. Selecting the view location ends the **Base View** tool.

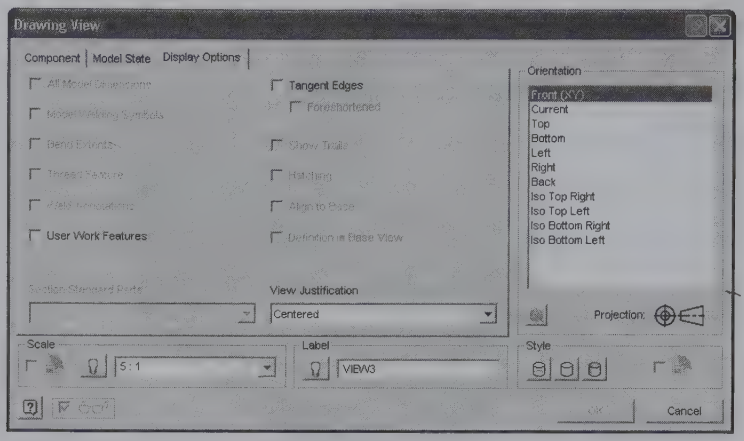


Exercise 17-1 Complete the Exercise on the Student CD.



Exercise 17-2 Complete the Exercise on the Student CD.

Figure 17-10.
The **Drawing View** dialog box, **Display Options** tab.



Working with Base Views

Once you create a base drawing view, there are a variety of options available for manipulating view characteristics. Base views are free to move anywhere in two-dimensional space. To move a view, hover your cursor over the view boundary until you see the **Move** icon. Then, hold down the left mouse button, drag the view to the desired location, and release the left mouse button. See **Figure 17-11.**

There are multiple reasons for selecting drawing geometry. For example, you may want to hide the visibility of a certain object, or override specific object parameters. To select one object, pick the item using your left mouse button. To select multiple objects, apply one of the following techniques:

- Hold down the [Ctrl] key and pick the desired number of objects.
- Use a crossing selection to select all objects that come in contact with the crossing selection box. To create a crossing selection, first hold down the left mouse button at a point inside or outside of the view boundary starting from the upper- or lower-right side of the geometry you want to select. Then, drag the corner of the crossing window diagonally across the object you want to select, and release the left mouse button. See **Figure 17-12.**

Figure 17-11.
Moving a drawing view.

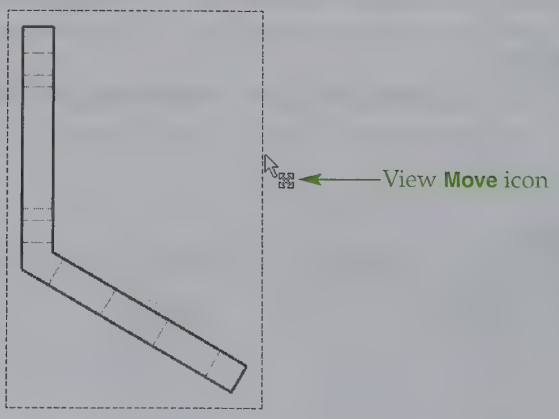
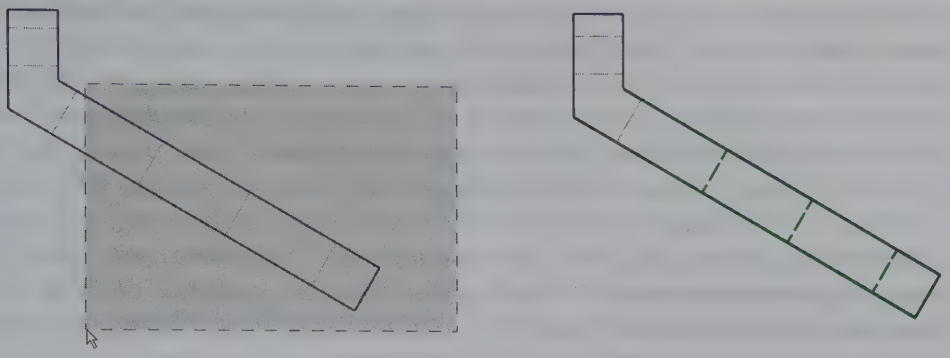


Figure 17-12.
Using a crossing selection to pick multiple drawing objects.



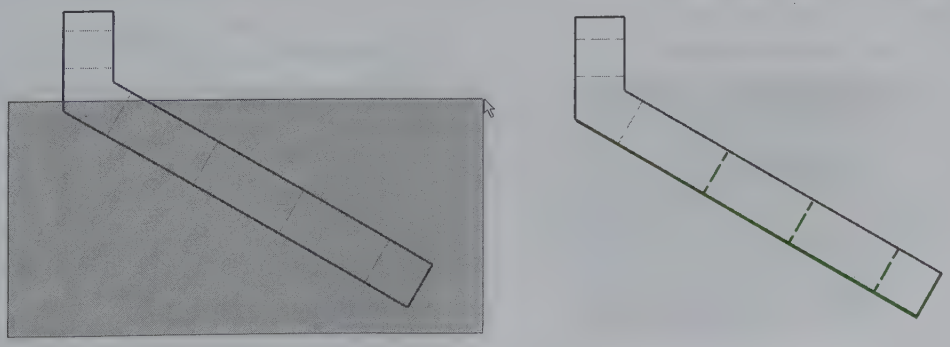
- Use a window selection to select all objects that are entirely inside the window selection. To create a window selection, first hold down the left mouse button at a point inside or outside of the view boundary starting from the upper- or lower-left side of the geometry you want to select. Then, drag the corner of the crossing window diagonally across the object you want to select, ensuring that all objects are fully inside the window. Finally, release the left mouse button. See **Figure 17-13.**

Use the **Measure Distance** and **Measure Angle** options available from the **Tools** pull-down menu to measure drawing view geometry. When you access the **Measure Distance** tool, the **Measure Distance** dialog box is displayed, which allows you to select and display the length of curves and location of points. When you access the **Measure Angle** tool, the **Measure Angle** dialog box is displayed, which allows you to select and display the angle between two lines.

Many drawing view characteristics can be manipulated by right-clicking inside a view boundary, or on the view name in the browser, and selecting one of the following options:

- **Open.** Pick this option to automatically open the model file used to generate the selected drawing view.
- **Edit View...** Choose this option to reopen the **Drawing View** dialog box,. The **Drawing View** dialog box allows you to edit many of the view options specified when the view was created.
- **Rotate.** Choose this option to access the **Rotate View** dialog box, as shown in **Figure 17-14.** The **Rotate View** dialog box allows you to rotate the selected view.

Figure 17-13.
Using a window selection to pick multiple drawing objects.



The first step is to specify how you want to rotate the view by selecting **Edge** or **Angle** from the **By** drop-down list. The **Edge** option rotates the view by aligning a selected edge horizontally or vertically. To use the **Edge** rotation technique, pick the edge you want to align, and then pick the **Horizontal** or **Vertical** radio button, depending on the application. For example, if you want an edge that is currently horizontal to become vertical, pick the horizontal edge and the **Vertical** radio button. Next, select the **Counterclockwise** button to rotate the view counterclockwise, or the **Clockwise** button to rotate the view clockwise. When you are satisfied with the view rotation, pick the **OK** button to exit the tool. The **Angle** option rotates the view at a specified angle. To use the **Angle** rotation technique, enter the rotation angle in the **Angle** edit box. Next, select the **Counterclockwise** button to rotate the view counterclockwise, or the **Clockwise** button to rotate the view clockwise. When you are satisfied with the view rotation, pick the **OK** button to exit the tool.

NOTE



The position of annotations, such as dimensions, is automatically modified to remain associated to the view geometry when you rotate drawing views.

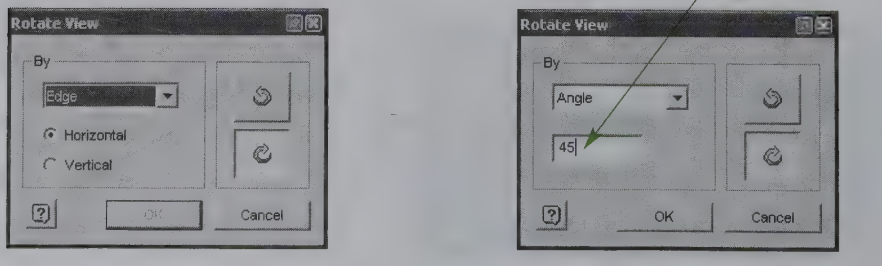
- **Annotation Visibility.** This cascading submenu controls the visibility of model and drawing annotations. Model dimensions are parametrically controlled dimensions related directly to the model, while drawing dimensions are additional dimensions specified after a view is created. The following cascading submenu options may be available depending on the type of drawing view (part, assembly, weldment, or presentation):
  - **Model Dimensions.** Pick this option to display model dimensions in the specified view.
  - **Drawing Dimensions.** Select this option to display drawing dimensions in the specified view.
  - **Model Weld Symbols.** Pick this option to display weld symbols applied to a weldment.
  - **Weld Annotations.** Pick this option to display weld annotations, such as caterpillars and end treatments, applied to a weldment.

NOTE



You must obtain annotations from the model, or add drawing annotations, in order to turn the visibility of the annotations on and off.

Figure 17-14.
Views can be rotated by an edge or an angle, as shown in these **Rotate View** dialog boxes.



- **Automated Centerlines...** Select this option to access the **Automated Centerlines** dialog box. Again, the **Automated Centerlines** dialog box is used to specify which drawing items automatically receive centerlines and center marks. However, when you access this dialog box from an individual drawing view, the specified centerlines and center marks are only added to the selected view.
- **Retrieve Dimension...** Select this option to access the **Retrieve Dimension** dialog box, which allows you to add model dimensions to selected drawing view objects. However, if you select the **Retrieve all model dimension on view placement** check box in the **Drawing** tab of the **Options** dialog box, model dimension are automatically displayed.
- **Get Model Annotations.** This cascading submenu allows you to add weldment model annotations to the drawing view. Select the **Get Welding Symbols** option to add weldment weld symbols, and pick the **Get Weld Annotations** option to display weld annotations, such as caterpillars and end treatments, applied to a weldment.

NOTE



Once you obtain model annotations, use the **Annotation Visibility** cascading submenu to control the visibility of the annotations.

PROFESSIONAL TIP



You can edit the properties of specific drawing features in a view by right-clicking the component in the browser and expanding the view to display the model contents. Then, pick the **Properties...** menu option. You can also right-click on model features in the view, and pick the **Properties...** menu option.

Projecting Views

Once you create a base drawing view, such as a front view, using the **Base View** tool, the next step in creating a multiview drawing, depending on the application, is to project additional views, such as a top and right side view. Projected views are generated quickly and easily from existing base views or dependent views, because model information is already stored in the drawing. The **Projected View** tool allows you to project as many views as needed from base or dependent views, assuming at least one view, referencing a model, exists in the drawing. You can use the **Projected View** tool anytime during drawing development and generate a wide variety of additional model views. To access the **Projected View** tool pick the **Projected View** button on the **Drawing Views Panel** bar; pick the **Projected View** button on the **Drawing Views Panel** toolbar; right-click an existing parametric drawing view in the graphics window, and select **Projected** from the **Create View** cascading submenu; right-click an existing parametric drawing view name in the browser, and select **Projected** from the **Create View** cascading submenu; or from the **Insert** pull-down menu, select the **Projected View** option of the **Model Views** cascading submenu.

NOTE

If you access the **Projected View** tool from the **Drawing Views Panel** bar, **Drawing Views Panel** toolbar, or the **Insert** pull-down menu, you must select the view from which you want to project.

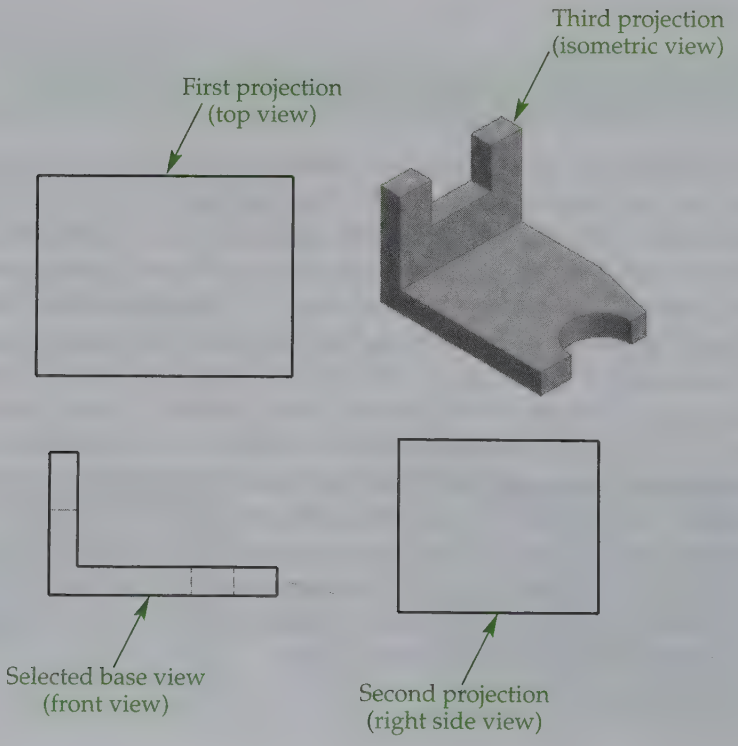
When you access the **Projected View** tool and a view is selected, use your mouse to move the projected view in place. See **Figure 17-15**. When you are satisfied with the location of the projected view, select the location by pressing the left mouse button. Continue placing views as desired using the techniques described. You only need to access the **Projected View** tool once to create multiple views. Once you are satisfied with the number of projected views, right-click and select the **Create** menu option to generate the projected views.

NOTE

Each of the view editing display options in the **Base View** section applies to editing projected views.

After you create a projected view, you can move the view by picking inside the view boundaries and dragging the view to the desired location. However, you do not have total control over the movement of a projected view, because of alignment constraints with the base view. Similarly, when you move the base view, projected views also move because of alignment constraints. Typically, the alignment constraints associated with drawings ensure valid relationships between views and do not need to be edited. However, you can modify the alignment of views by right-clicking inside

Figure 17-15.
The first and second projected views have been placed, and the third projected view shows a preview because it has not been placed yet.



the view boundaries and selecting one of the alignment options from the **Alignment** cascading submenu; right-clicking on the view name in the browser, and selecting one of the alignment options from the **Alignment** cascading submenu; or picking the view you want to align, then from the **Tools** pull-down menu, select an alignment option from the **Align Views** cascading submenu.



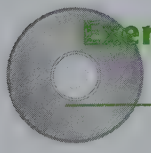
NOTE If an alignment constraint already exists, you will not be able to modify alignment of the view, unless the constraint is removed using the **Break** alignment option.

Four alignment options are available. Select the **Horizontal** option to create a horizontal alignment constraint, which allows you to move the view only horizontally, or along the X axis. Choose the **Vertical** option to create a vertical alignment constraint, which allows you to move the view only vertically, or along the Y axis. Select the **In Position** option to create a constraint that is neither vertical nor horizontal. For example, this type of alignment is typically used for projected isometric views, which are not constrained to the X or Y axis of a base view. Pick the **Break** option to remove alignment constraints. A view with a broken alignment can be moved anywhere, without being controlled by another view. You must first break the existing alignment constraints of a view, before the alignment can be modified.

Once the view you want to align is selected and you have chosen the type of alignment you want to create (horizontal, vertical, in position, or break), the final step is to pick the view from which you want the alignment to be referenced, which is typically the base view.

Deleting Dependent Views

Any base view that has a dependent view, such as a projected or auxiliary (covered later) view, cannot be deleted until you decide whether the dependent view should also be deleted. When you right-click a view and select **Delete**, one of two different windows appears. See Figure 17-16. If the selected view has one or more dependent views, you can choose which dependent views should also be deleted by picking the **More** button. A list of dependent views appears with the word Yes next to each, indicating that the dependent views will be deleted. See Figure 17-17. Pick Yes and it changes to No, preventing the dependent view from being deleted.



Exercise 17-3 Complete the Exercise on the Student CD.

Figure 17-16.
One of these windows will appear when you delete a view.

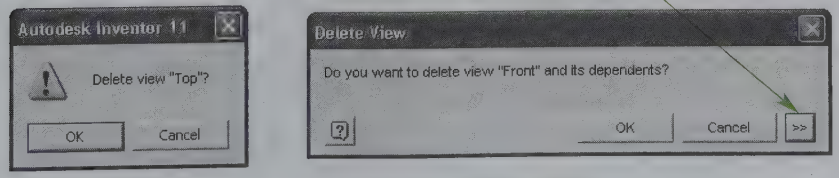
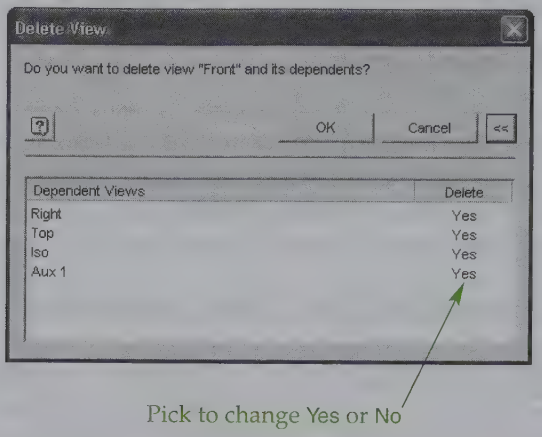


Figure 17-17.
If the view you are deleting has dependent views, you can decide if they should be deleted as well.

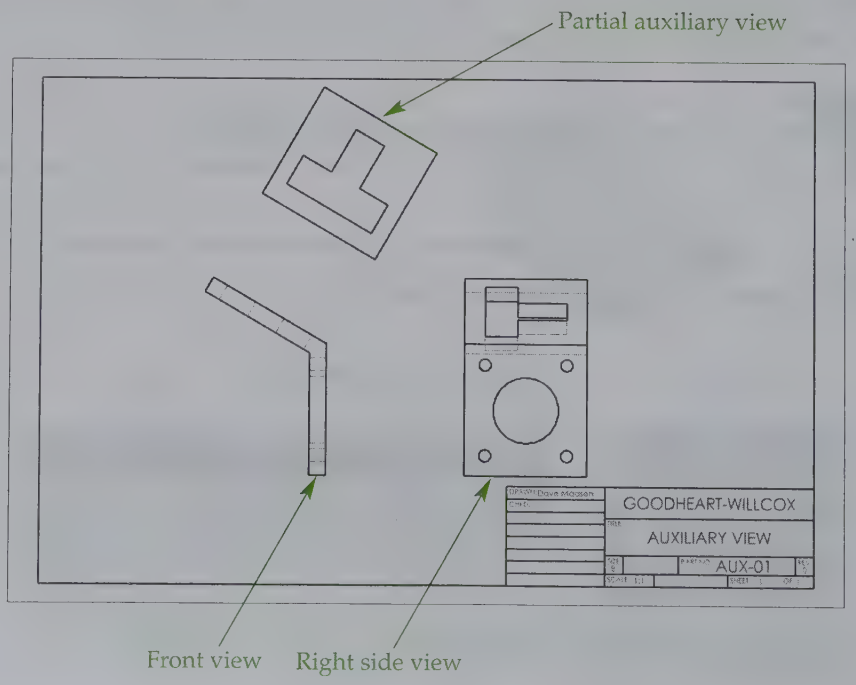


Creating Auxiliary Views

Auxiliary views are used to show the true size and shape of an inclined surface that is not parallel to any of the projected views, including the front, top, bottom, left and right side, and back views. See Figure 17-18.

Like projected views, auxiliary views are generated from existing base views or dependent views, because model information is already stored in the drawing. The **Auxiliary View** tool allows you to create auxiliary views, which are projected from existing inclined surfaces onto the auxiliary plane. Assuming at least one view referencing a model exists in the drawing; you can use the **Auxiliary View** tool anytime during drawing development. To access the **Auxiliary View** tool pick the **Auxiliary View** button on the **Drawing Views Panel** bar; pick the **Auxiliary View** button on the **Drawing Views Panel** toolbar; right-click an existing

Figure 17-18.
An example of a partial auxiliary view.



parametric drawing view in the graphics window, and select **Auxiliary** from the **Create View** cascading submenu; right-click an existing parametric drawing view name in the browser, and select **Auxiliary** from the **Create View** cascading submenu; or from the **Insert** pull-down menu, select the **Auxiliary View** option of the **Model Views** cascading submenu.

NOTE



If you access the **Auxiliary View** tool from the **Drawing Views Panel** bar, **Drawing Views Panel** toolbar, or the **Insert** pull-down menu, you must select the view you want to project from.

When you access the **Auxiliary View** tool and a view is selected, the **Auxiliary View** dialog box is displayed. See **Figure 17-19.** Use the **Label** area to modify the view label. You can edit the default name of the label in the **Label** edit box and pick the **Toggle Label Visibility** button if you want the label displayed in the view. The **Scale** area is used to define the scale of the view by entering a scale in the **Scale** edit box, or select a scale from the flyout button. If you want the view scale to be displayed in the view, select the **Toggle Scale Visibility** button. The **Style** area allows you to specify the display of the view. You can choose to display the view with hidden lines by picking the **Hidden Lines** button, without hidden lines by picking the **Hidden Lines Removed** button, or as shaded by picking the **Shaded** button.

Once you specify the label, scale and display options in the **Auxiliary View** dialog box, pick the edge of the surface from which you want the auxiliary view to be projected. The selected edge can be parallel or perpendicular to the desired auxiliary view. Next, drag the preview of the auxiliary view to the desired location. See **Figure 17-20.** Finally, generate the auxiliary view by selecting the location where you want the view to be displayed, or by picking the **OK** button in the **Auxiliary View** dialog box.

If the auxiliary view you created is not acceptable, you can redefine the surface edge by right-clicking inside the auxiliary view boundaries and selecting the **Realign auxiliary views** menu option, or by right-clicking on the auxiliary view name in the browser, and selecting the **Realign auxiliary views** menu option. Once you access the **Realign auxiliary views** tool, select the edge you want to use to redefine the auxiliary view, and pick the location where you want the new view to be displayed.

Often, *partial auxiliary views* are used to show the true size and shape of only the inclined surface, eliminating any projected geometry that may be foreshortened. See **Figure 17-21.** There are no tools in Inventor that allow you to create a partial auxiliary view directly, unless you want to create an auxiliary view section. However, partial auxiliary views can be drawn by turning off the visibility of specific geometry. To

Figure 17-19.
The **Auxiliary View** dialog box.

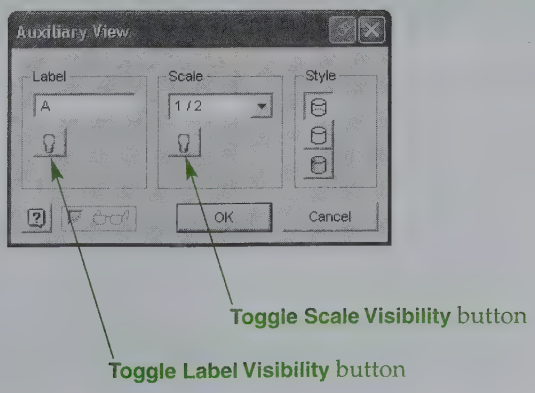


Figure 17-20.
After selecting an auxiliary view edge, the auxiliary view preview appears during the view placement.

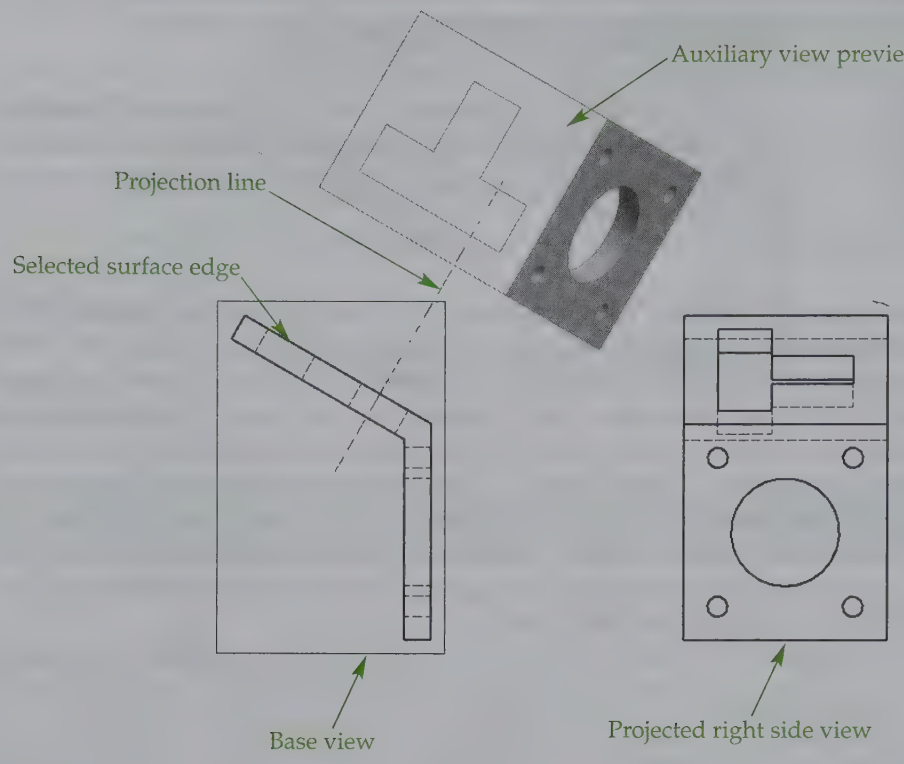
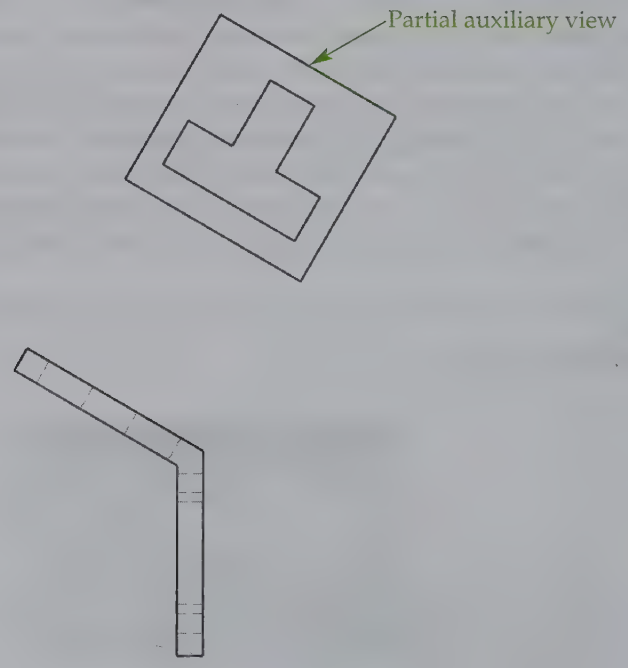


Figure 17-21.
An example of a partial auxiliary view.



disable geometry visibility, right-click the object or objects you want to hide and deselect the **Visibility** pop-up menu option. See Figure 17-22. Continue turning off the visibility of geometry until the partial view is created.

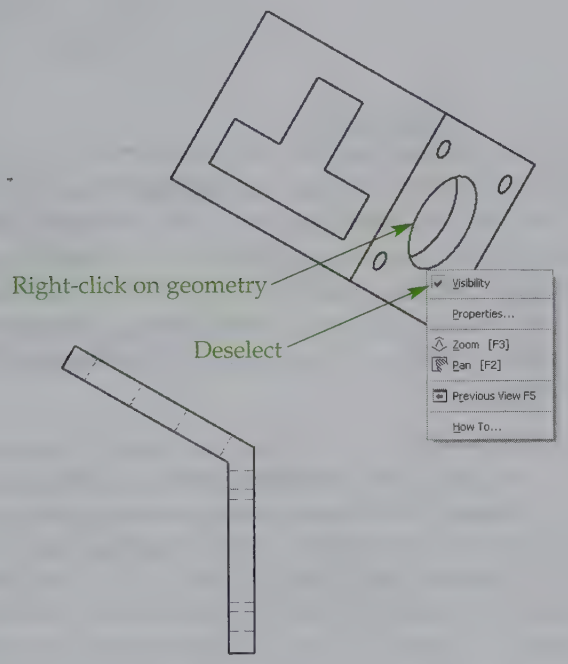
NOTE

You can disable the visibility of most geometry in all types of views, not just auxiliary views.

Exercise 17-4

Complete the Exercise on the Student CD.

Figure 17-22.
Creating a partial auxiliary view.

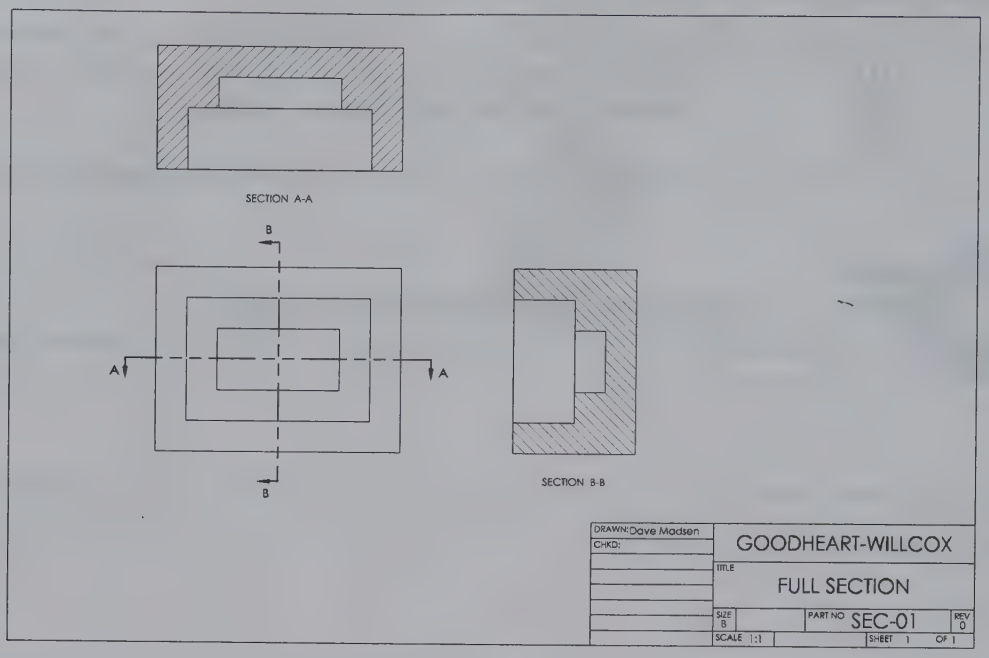


Developing Section Views

Many part models have complex interior geometry that is difficult to visualize and display in a drawing. To overcome this problem, ***sectional views***, also known as ***section views*** or ***sections***, are used to expose the interior features of a part. Typically, sectioning a view eliminates hidden lines, which allows you to dimension geometry and provide a much clearer part representation. See Figure 17-23.

Like projected views, section views are generated from existing base or dependent views, because model information is already stored in the drawing. The **Section View** tool allows you to create section views, which are projected from a specified cutting plane, through an existing surface. The **Section View** tool can be used anytime during

Figure 17-23.
An example of section views, full sections.



drawing development, assuming at least one view referencing a model exists in the drawing. To access the **Section View** tool pick the **Section View** button on the **Drawing Views Panel** bar; pick the **Section View** button on the **Drawing Views Panel** toolbar; right-click on an existing parametric drawing view in the graphics window, and select **Section** from the **Create View** cascading submenu; right-click on an existing parametric drawing view name in the browser, and select **Section** from the **Create View** cascading submenu; or from the **Insert** pull-down menu, select the **Section View** option of the **Model Views** cascading submenu.

NOTE



If you access the **Section View** tool from the **Drawing Views Panel** bar, **Drawing Views Panel** toolbar, or the **Insert** pull-down menu, you must select the view from which you want to project the section.

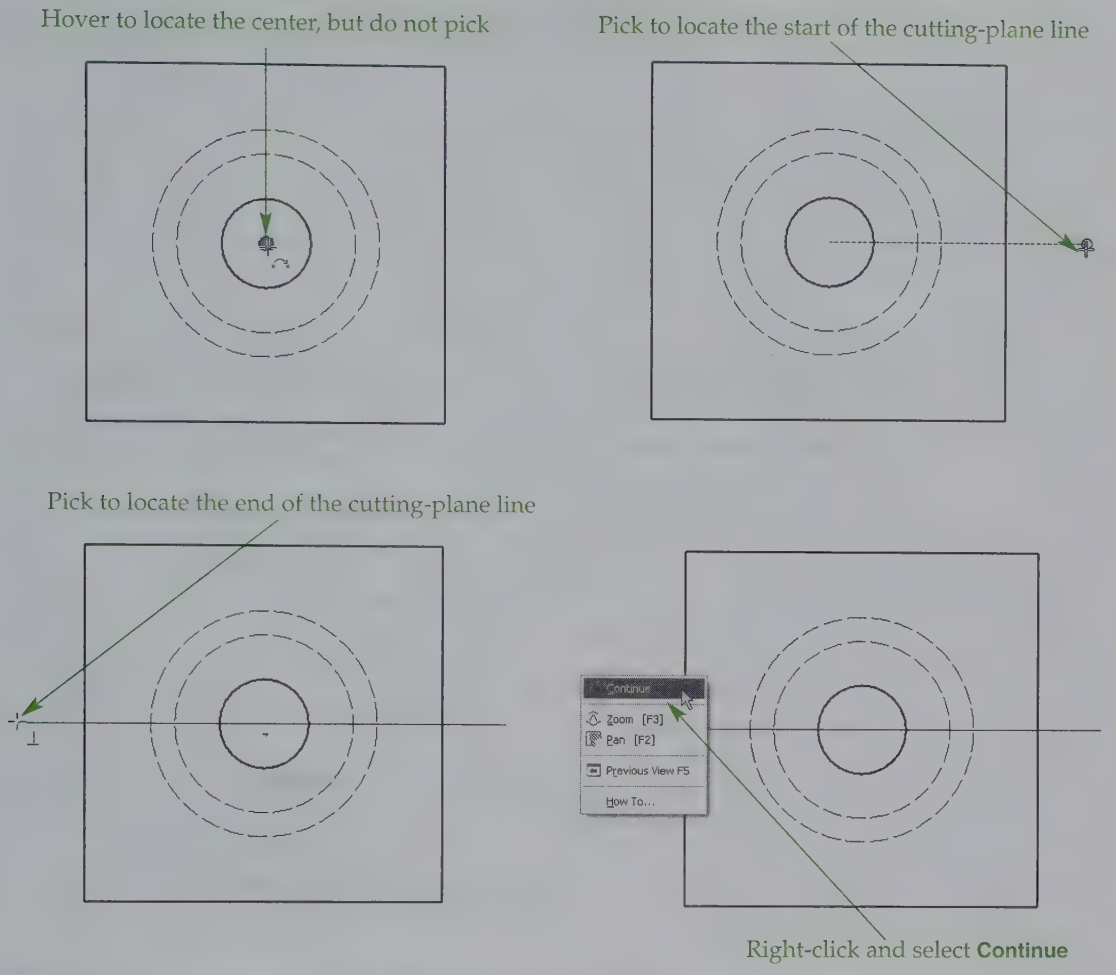
Cutting-Plane Lines

When you access the **Section View** tool and a view is selected, the next step in creating a section view is to define a cutting-plane line. The ***cutting-plane line*** represents the cutting plane of the section, which is the location where the view is sliced. Placing a cutting-plane line is very similar to sketching a line in the Inventor 2D sketch environment.

Full section

A ***full section*** shows the entire plane inside a part, and is created from a single cutting-plane line. If you want the cutting plane to go through the center of an object, hover over the midpoint of a line or the center of a circle until a green dot appears. Now, move the cursor to the point where the cutting-plane line will start, and pick when the alignment line appears. To place the end of the cutting-plane line, move the cursor to the other side of the part and pick when the cutting-plane line preview is horizontal. Right-click and select **Continue**. See Figure 17-24.

Figure 17-24.
Sketching a full-section cutting-plane line.



Half section

A ***half section*** is used when an area of a part does not have to be sectioned, because the area has no features, or the features are symmetrical to the sectioned features. The cutting-plane line of a half section is similar to that of a full section, with a 90° turn in the cutting-plane line. To create a half section, begin creating the cutting-plane line as described above, but place the second pick where you want the cutting-plane line to turn. Now move the cursor to create a 90° line from the first portion of the cutting-plane line and pick to place the end of the cutting-plane line. Right-click and select **Continue**. See Figure 17-25.

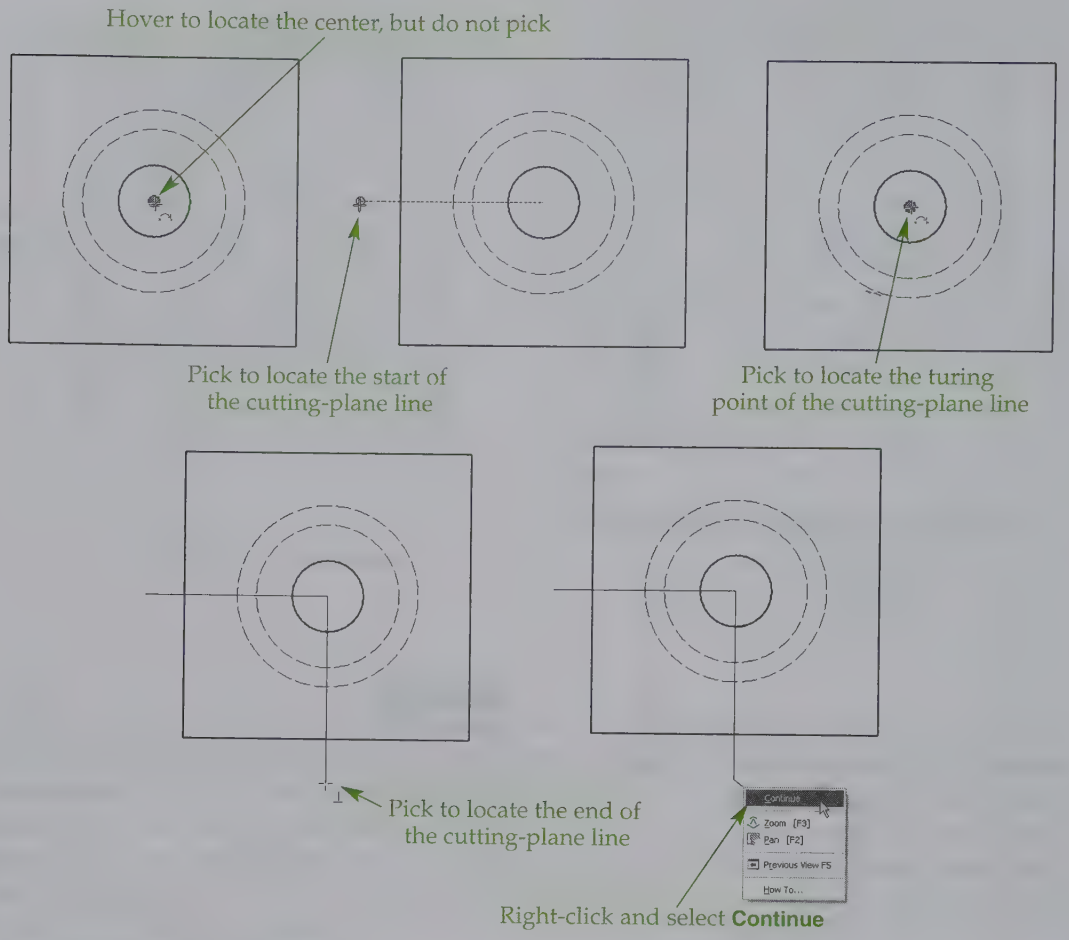
Offset section

An ***offset section*** has a cutting-plane line that travels through various depths of a part. This type of section is used when the features of a part cannot be seen with a single straight cutting-plane line. Creating an offset section requires the same steps as a half section; however, additional turns are placed at the depth of each feature. See Figure 17-26.

Aligned section

An ***aligned section*** has a cutting-plane line that changes directions at an angle other than 90°. This type of section is typically used when the cutting-plane line must

Figure 17-25.
Sketching a half-section cutting-plane line.

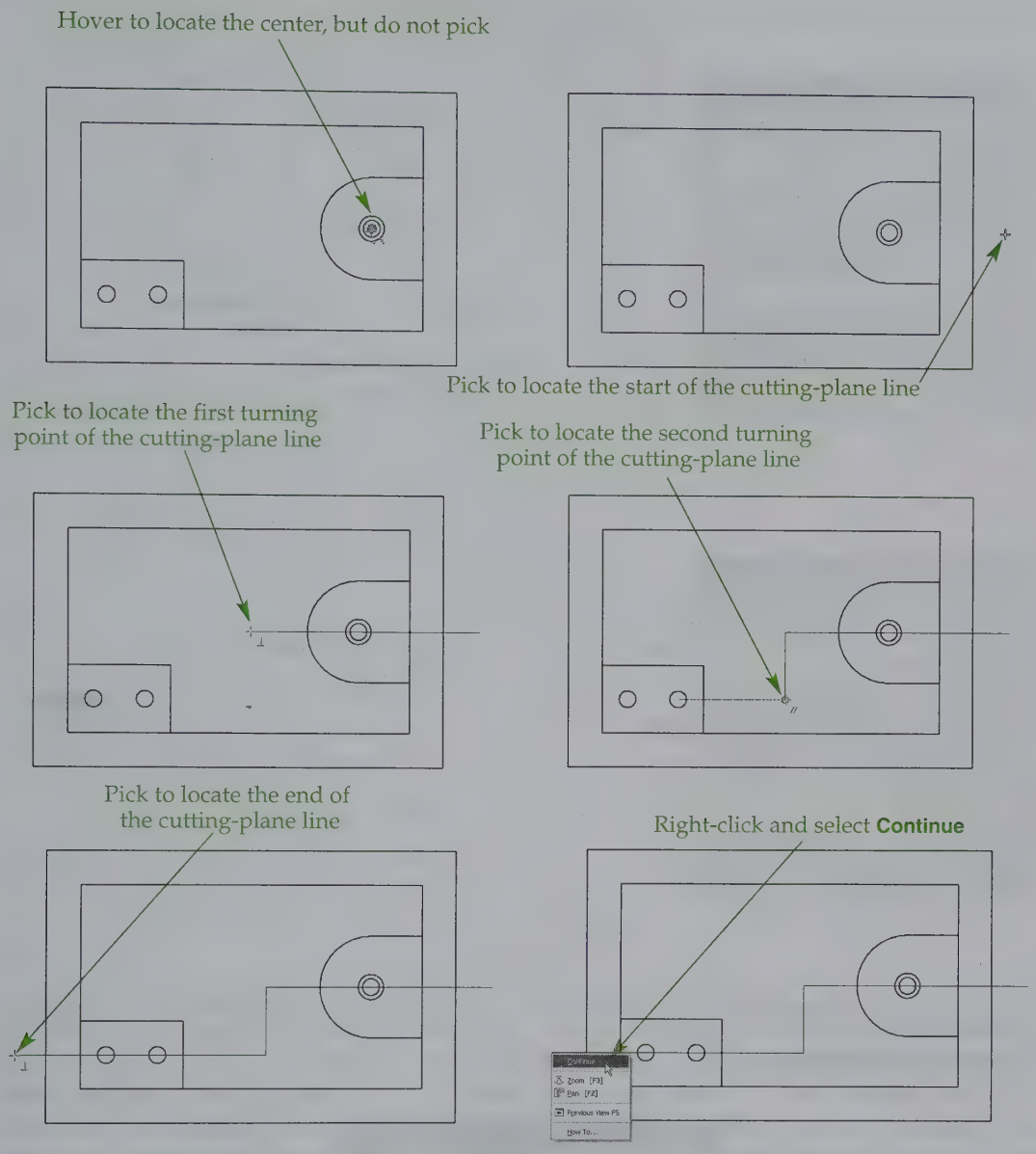


go through features that are revolved around the center of a cylindrical part. The top view should be oriented so that one of the features is horizontally in line with the center of the part. The cutting-plane line is built by aligning it with the center of the features. See Figure 17-27.

Once you sketch the cutting-plane line, right-click and select the **Continue** menu option. When you select **Continue**, the cutting-plane line is created, and the **Section View** dialog box is displayed. See Figure 17-28. Use the **Label** area to modify the view label. You can edit the default name of the label in the **Label** edit box and pick the **Toggle Label Visibility** button if you want the label displayed in the view. The **Scale** area is used to define the scale of the view by entering a scale in the **Scale** edit box, or select a scale from the flyout button. If you want the view scale to be displayed in the view, select the **Toggle Scale Visibility** button. The **Style** area allows you to specify the display of the view. You can choose to display the view with hidden lines by picking the **Hidden Lines** button, without hidden lines by picking the **Hidden Lines Removed** button, or as shaded by picking the **Shaded** button.

There are two options for defining how much of a view is removed during the section view operation. Picking the **Full** section depth option from the **Section Depth** area is effective for most sectioning applications, and removes all of the material from the cutting-plane line out. The second section depth option involves selecting the **Distance** choice from the **Section Depth** area and entering portion of material retained in the **Distance** edit box. The **Distance** option is not a typical sectioning operation, and generates a portion of

Figure 17-26.
Sketching an offset-section cutting-plane line.



material as opposed to an actual section view. **Figure 17-29** shows an example of the two section depth options.

After you specify the label, scale, display options, and section depth in the **Section View** dialog box, drag the preview of the section view to the desired location. See **Figure 17-30**. Finally, generate the section view by selecting the location where you want the view to be displayed or by picking the **OK** button in the **Section View** dialog box.

An alternate way to create a section view is to sketch a cutting-plane line using sketch tools and then access the **Section View** tool. To use this technique, you must first associate the sketch with the drawing view to which you want to section. You can associate a sketch with a view by first picking the view and then opening a new sketch. Sketching a cutting-plane line is just like sketching any other two-dimensional geometry. For more information on 2D sketching, refer to previous chapters and the topics on specific drawing environment sketch tools. Additionally, a pre-drawn cutting-plane

Figure 17-27.
Sketching an aligned-section cutting-plane line.

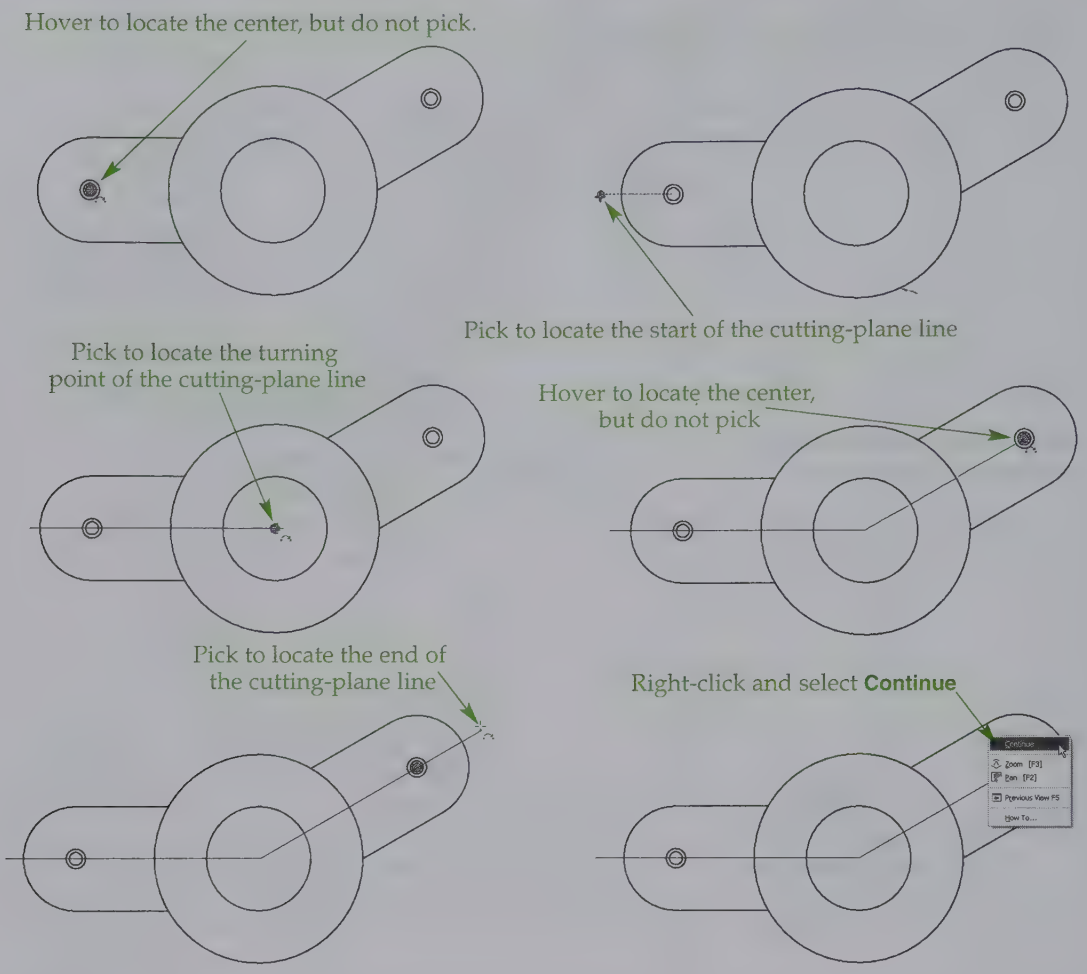


Figure 17-28.
The **Section View** dialog box.

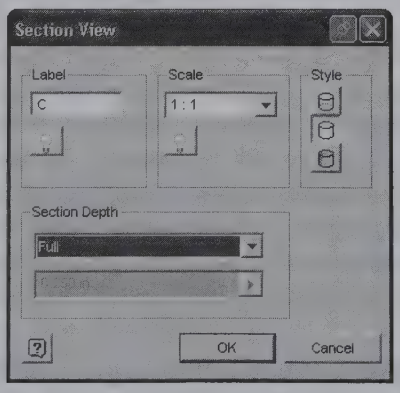
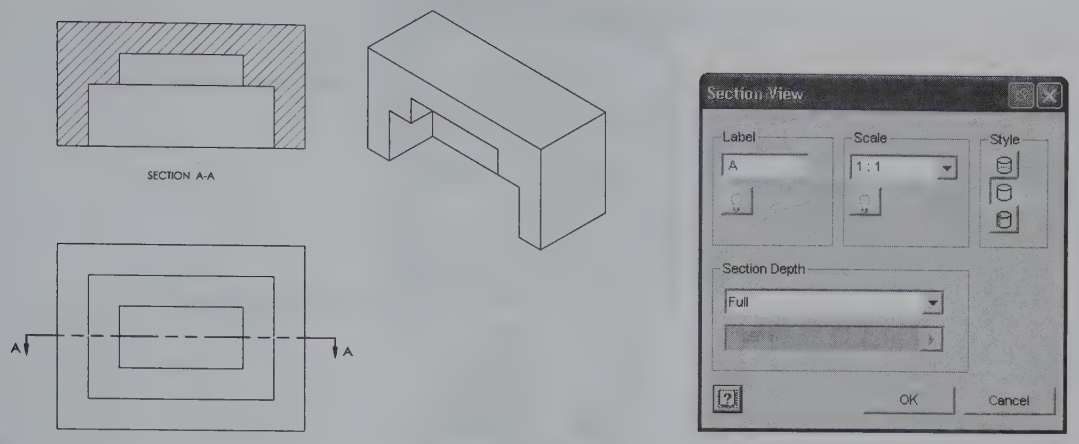
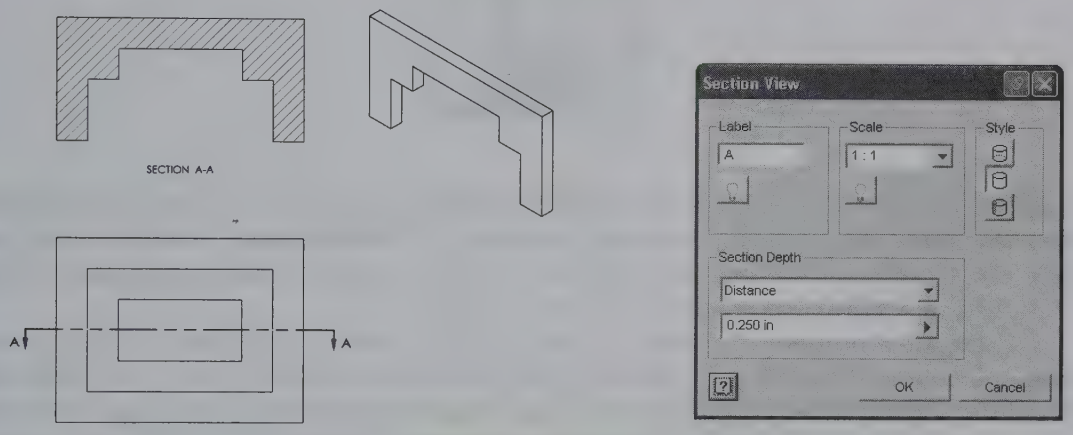


Figure 17-29.
A section view created using the **Full section depth** option and the **Distance section** option.



Full section depth



Distance section depth

line sketch can only be used to create a section view if you access the **Section View** tool by right-clicking the sketch name in the browser, and selecting the **Create Section View...** pop-up menu option, or by picking the sketch name in the browser, and then selecting the **Section View** tool from the **Drawing Views Panel** toolbar, **Drawing Views Panel** bar, or **Insert** pull-down menu. When you access the **Section View** tool using one of the previous methods, the sketch is automatically selected, and the **Section View** dialog box is displayed.

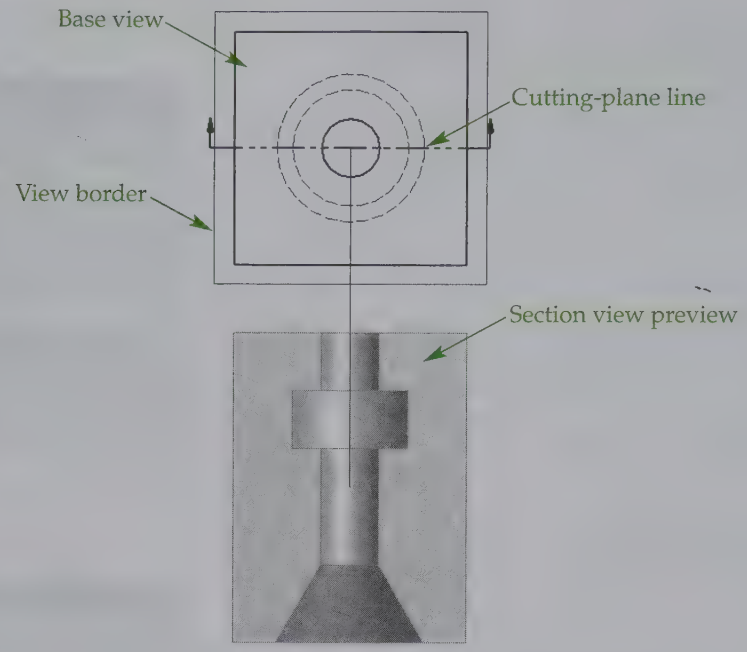
NOTE



You must associate a sketch with a drawing view in order to project and use drawing geometry in the sketch.

You will notice that the section lines, cutting-plane lines, and labels are automatically created, based on your specifications. However, section lines, the cutting-plane line, and the section view label can be modified, if necessary. ***Section lines*** show where the section has cut through the material. The default section lines displayed when you create a section view correspond to hatch style settings in the **Styles and Standards Editor**. To change the display of section lines in a section view, right-click on the hatching and pick the **Modify Hatch...** menu option to open the **Modify Hatch Pattern** dialog

Figure 17-30.
The section view preview.



box. See Figure 17-31. Refer to the previous content about hatch styles and the **Styles and Standards Editor** for information regarding the **Modify Hatch Pattern** dialog box options. To turn off section line visibility, right-click on the hatching in the section view and select the **Hide Hatch** pop-up menu option.

There are a number of options for modifying a cutting-plane line. To change the size or location of a cutting-plane line, right-click on the cutting-plane line or the section view sketch listed in the browser, and select the **Edit** menu option to enter the sketch environment. Cutting-plane lines are sketched features, and are edited like any other sketch lines. Once you change the size or location of the sketch, exit or finish the sketch to complete the modification. In addition to changing the actual physical size or location of a cutting plane line, you can reverse the cutting-plane direction arrows by right-clicking on a cutting-plane line and selecting the **Reverse Direction** menu option. Also, many sections in which the extents of the cutting-plane are obvious do not contain full-length cutting-plane lines, as shown in Figure 17-32. To remove the inner portion of a cutting-plane line, right-click on the cutting-plane line and deselect the **Show Entire Line** menu option. Finally, you have the option of editing the section depth by right-clicking on a cutting-plane line and picking the **Edit Section Depth** menu option. Selecting **Edit Section Depth** opens the **Edit Section Depth** dialog box, which is used exactly like the **Section Depth** area of the **Section View** dialog box.

You can edit the default label parameters by right-clicking inside the section view boundaries or the section view name in the browser and selecting the **Edit View...** menu option. However, you can modify and add information to the label text by right-clicking the label and selecting the **Edit View Label...** menu option to display the **Format Text** dialog box.



Exercise 17-5 Complete the Exercise on the Student CD.

Figure 17-31.
The **Modify Hatch Pattern** dialog box.

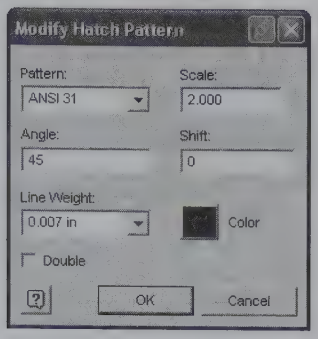
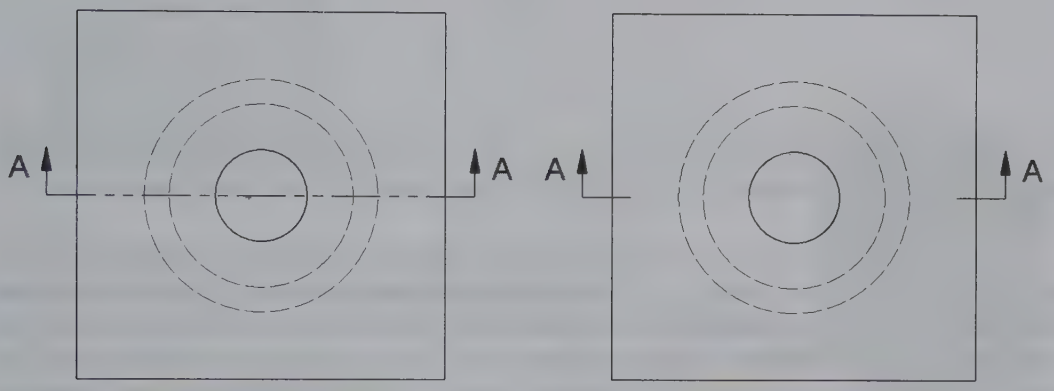


Figure 17-32.
A cutting-plane line can be full-length or just the ends, depending on status of the **Show Entire Line** check box.



Show Entire Line is on

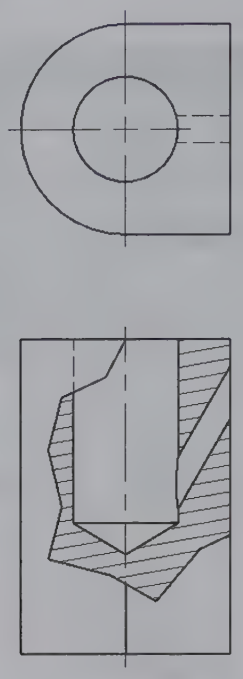
Show Entire Line is off

Developing Broken-out Views

Just as with full, half, offset, aligned, and other section views, ***broken-out section views***, also know as ***broken-out sections***, are used to expose the interior features of a component, allowing you to better visualize and display complex interior geometry. However, broken-out sections remove, or section, only a small portion of specified drawing view geometry without using typical cutting plane lines. Again, in most cases, hidden lines are omitted in a broken-out section view, which allows you to dimension geometry and display a much clearer component representation. See Figure 17-33.

The **Break Out View** tool allows you to create broken-out section views, which are projected from a specified cutting plane, through an existing surface. Before you use the **Break Out View** tool, you must sketch a broken-out section profile associated with the view. The sketch profile defines the location and extent of the broken-out section. The section profile must be associated with the view or you will not be able to create a broken-out section. To create a sketch that is associated with the view, you must first pick the view. A view is selected when the dotted view boundary line appears and the view is highlighted in the browser. Next, with the view selected, open a new sketch by picking the **2D Sketch** button on the **Inventor Standard** toolbar, or by pressing the [S] key. Sketching a broken-out section profile is just like sketching other 2D geometry. For

Figure 17-33.
An example of a broken-out section view.

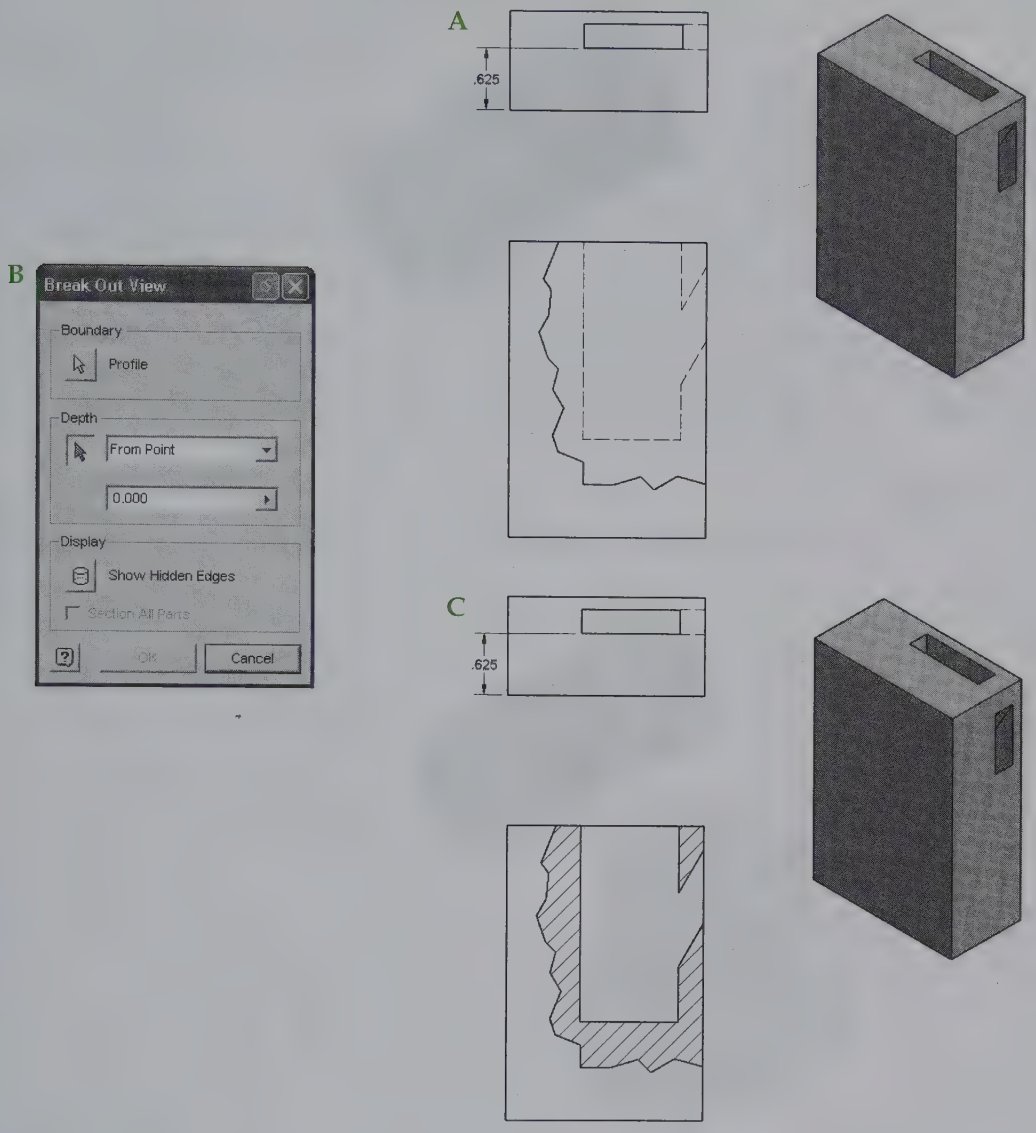


more information on 2D sketching, refer to previous chapters and topics on specific drawing environment sketch tools. As shown in Figure 17-34A, a broken-out sketch profile must be closed.

Once you place drawing views, and sketch a broken-out section profile associated with the view you plan to section, access the **Broken Out View** tool by picking the **Break Out View** button on the **Drawing Views Panel** bar; picking the **Break Out View** button on the **Drawing Views Panel** toolbar; right-clicking on an existing parametric drawing view in the graphics window, or the drawing view name in the browser, and selecting **Break Out** from the **Create View** cascading sub-menu; from the **Insert** pull-down menu, select the **Break Out View** option of the **Model Views** cascading submenu; or right-click the sketch profile in the graphics window, or the sketch name in the browser, and select the **Create Break Out View...** pop-up menu option.

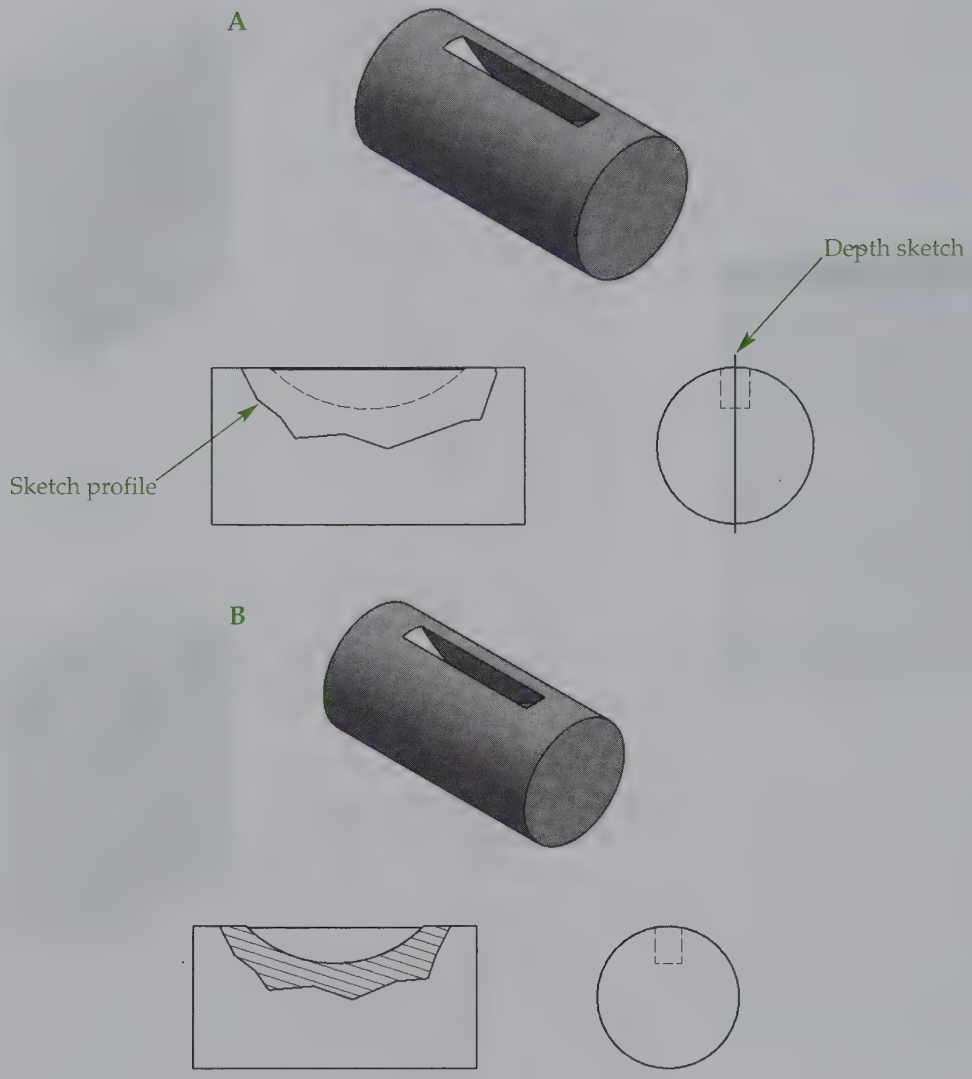
Inventor provides you with the four options for creating a broken-out section, depending on the desired section depth. Section depth defines where a view is sectioned, and how much material is removed. The **From Point** option allows you to create a broken-out section that is a specified distance away from a selected point on an existing base or projected drawing view. To use the **From Point** depth type, you must first create the view that you want to section and a broken-out section sketch profile associated with the view. Then, access the **Break Out View** tool, and pick the drawing view that contains the associated broken-out section sketch profile. When you pick the drawing view, the **Break Out View** dialog box is displayed. See Figure 17-34B. Initially the sketched profile may automatically be selected. If not, pick the **Sketch Profile** button, located in the **Boundary** area, and select the closed loop sketch profile. Next, to define a broken-out section using the **From Point** option, select **From Point** from the **Depth Type** drop-down list, and pick the **Select Depth** button. Then pick a point on the selected view, or a different drawing view. The point you select defines the location from which the section depth occurs. If you want to have temporary access to hidden lines that are not displayed, in order to select a point, pick the **Show Hidden Edges** button. Next, enter or select the section depth in the **Depth** edit box, and pick the **OK** button to generate the broken-out section view. Figure 17-34C shows the process of creating a broken-out section using a distance of .625″ from the selected point.

Figure 17-34.
A—An example of a broken-out section sketch profile. B—The **Break Out View** dialog box. C—Creating a **From Point** broken-out section view.



The **To Sketch** option allows you to create a broken-out section using a second sketch that is associated with a different drawing view. To use the **To Sketch** depth type, place an initial drawing view using a tool such as **Base View** or **Projected View**. Then, use the **Projected View** tool to project an additional drawing view. Choose the view you want to section, and sketch a broken-out section profile associated with the view. Finish the sketch, and then open another sketch associated with the other view, and use sketching tools to create a section depth. Sketching a broken-out section profile and depth is just like sketching any other 2D geometry. As shown in Figure 17-35A, a broken-out sketch profile must be closed, but a depth sketch can be open. Once you place drawing views, sketch a broken-out section profile associated with the view you plan to section, and sketch the section depth, access the **Break Out View** tool. Then pick the drawing view that contains the associated broken-out section sketch profile to display the **Break Out View** dialog box. See Figure 17-35B. Initially the sketched profile may be automatically selected. If not, pick the **Sketch Profile** button, located in the **Boundary** area, and select the closed loop sketch profile. Next, to define a broken-out section using the **To Sketch**

Figure 17-35.
A—An example of a broken-out section sketch profile and depth sketch. B—Creating a **To Sketch** broken-out section view.



option, select **To Sketch** from the **Depth Type** drop-down list, and pick the **Select Depth** button. Then pick the sketch geometry on the other view. Again, the sketch you select defines the location to which the section depth occurs. If you want to temporarily view hidden lines that are not displayed, pick the **Show Hidden Edges** button. Finally, pick the **OK** button to generate the broken-out section view. Figure 17-35B shows the process of creating a broken-out section using a sketch on a projected view.

The **To Hole** option allows you to create a broken-out section at the axis of a drawing view hole feature. To use the **To Hole** depth type you must first create the view that you want to section using a tool such as **Base View** or **Projected View**. Then, sketch a broken-out section profile associated with the view. Once you place drawing views and sketch a broken-out section profile associated with the view you plan to section, access the **Break Out View** tool, and then pick the drawing view that contains the associated broken-out section sketch profile. When you pick the drawing view, the **Break Out View** dialog box is displayed. See Figure 17-34B. Initially the sketched profile may be automatically selected. If not, pick the **Sketch Profile** button, located in the **Boundary** area, and select the closed loop sketch profile. Next, to define a broken-out

section using the **To Hole** option, select **To Hole** from the **Depth Type** drop-down list. Then pick a hole feature on the selected view, or a different drawing view. If you want to have temporary access to hidden lines that are not displayed, in order to select a hole, pick the **Show Hidden Edges** button. Finally, pick the **OK** button to generate the broken-out section view. Figure 17-36 shows the process of creating a broken-out section using a selected hole.

The **Through Part** option allows you to create a broken-out section using the thickness of a selected part to define the section depth. The process of creating a **Through Part** broken-out section view is similar to cut-extruding in the modeling environment. When you cut-extrude a sketch profile, material is removed to a specified point. However, with a **Through Part** broken-out section view, only the selected part is removed, allowing you to reveal interior parts in an assembly drawing. Though using the **Through Part** option allows you to cut through parts in assembly drawings, it is also useful when creating part drawings. For example, you may use a **Through Part** broken-out section view to generate a revolved section. See Figure 17-37. To use the **Through Part** depth type, you must first create the view that you want to section using a tool such as **Base View** or

Figure 17-36.
A sketch profile is created before creating a **To Hole** broken-out section view.

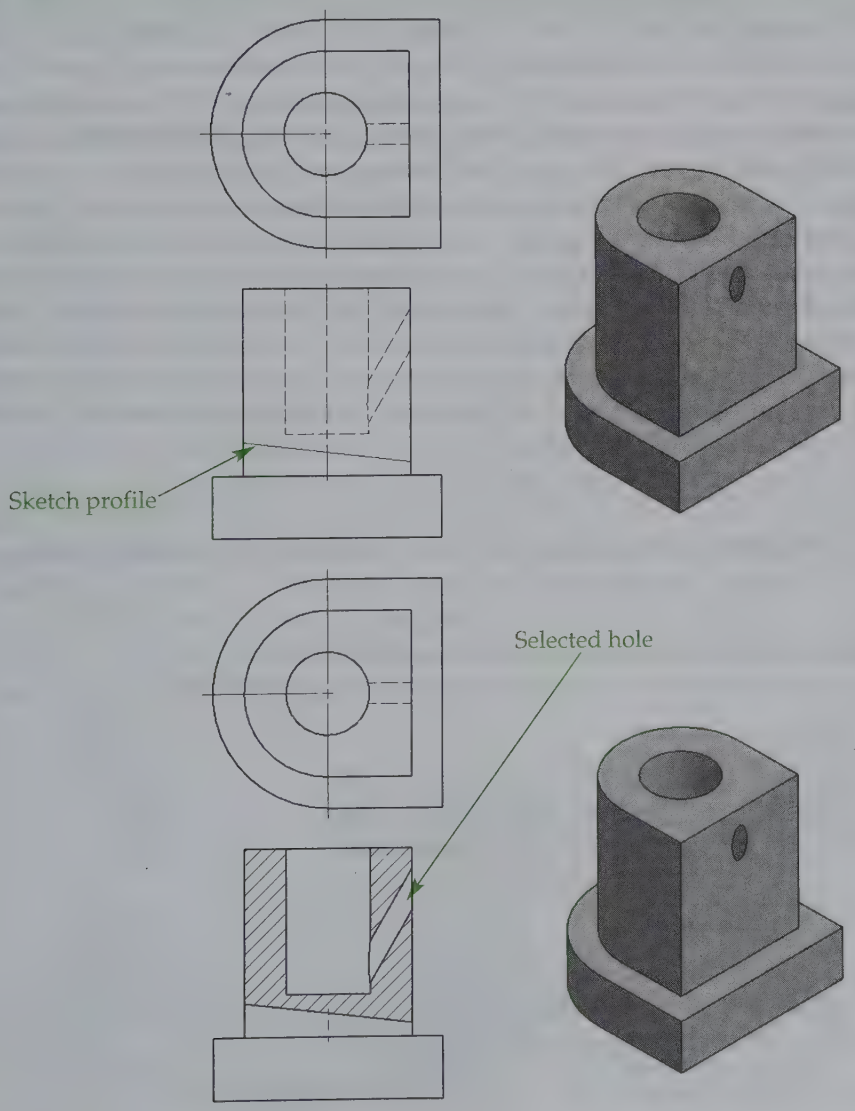
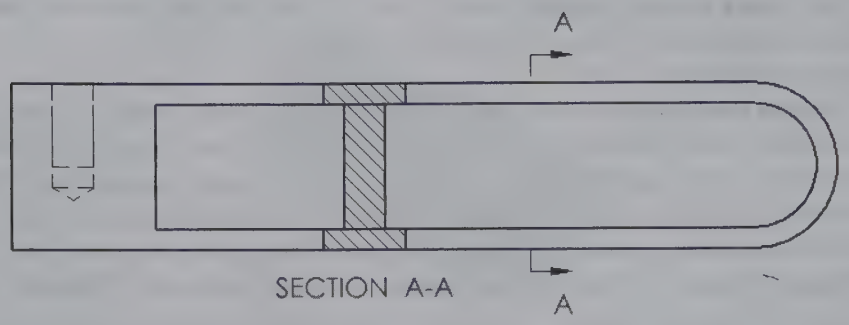


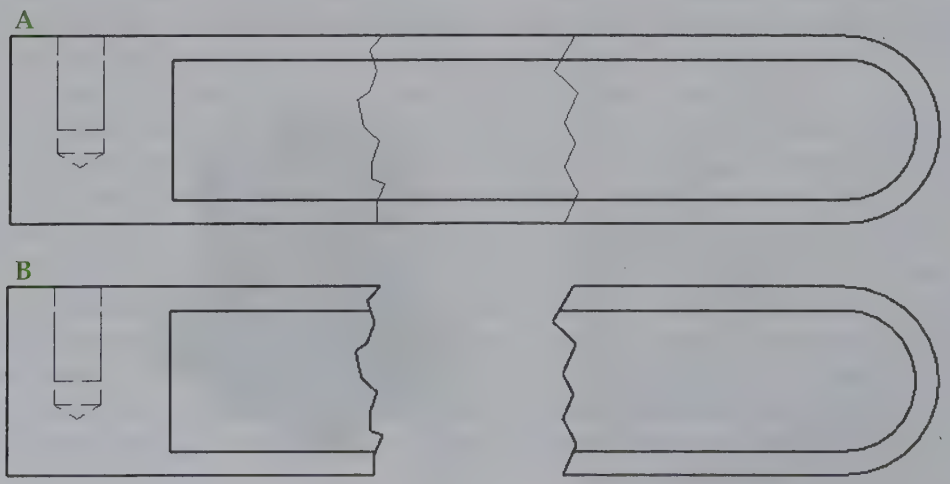
Figure 17-37.
A **Through Part** broken-out section view application, a revolved section.



Projected View. Then sketch a broken-out section profile associated with the view. As shown in Figure 17-38A, a broken-out sketch profile must be closed. Once you place drawing views and sketch a broken-out section profile associated with the view you plan to section, access the **Break Out View** tool, and then pick the drawing view that contains the associated broken-out section sketch profile. When you pick the drawing view, the **Break Out View** dialog box is displayed. See Figure 17-38B. Initially the sketched profile may be automatically selected. If not, pick the **Sketch Profile** button, located in the **Boundary** area, and select the closed loop sketch profile. Next, to define a broken-out section using the **Through Part** option, select **Through Part** from the **Depth Type** drop-down list. Then, pick the part through which you want to cut the sketched boundary. If you are working with a part drawing, only one part is selectable. If you want to have temporary access to hidden lines that are not displayed, pick the **Show Hidden Edges** button. Finally, pick the **OK** button to generate the broken-out section view.

When a broken-out section view is generated, the sketch or sketches used to create the view are consumed by the broken-out view. You can edit the sketch profile or a depth sketch anytime to modify the existing broken-out section view display. You can also reopen the **Break Out View** dialog box to edit the depth characteristics of the broken-out section by right-clicking on the break-out view in the browser, and selecting the **Edit Definition...** pop-up menu option. When a broken-out section view is created, section lines are automatically placed based on the hatch style settings in the

Figure 17-38.
A—An example of a broken-out section sketch profile. B—Creating a **Through Part** broken-out section view.



Styles and Standards Editor. However, you can modify section lines in a broken-out view by right-clicking on the hatching and picking the **Modify Hatch...** menu option, to open the **Modify Hatch Pattern** dialog box. Refer to the previous information about the **Hatch Style** area of the **Styles and Standards Editor**, or the section view content of this chapter for information regarding the options in the **Modify Hatch Pattern** dialog box. To turn off section line visibility, right-click on the hatching in the section view and select the **Hide Hatch** pop-up menu option.

NOTE

You must associate a sketch with a drawing view in order to project and use drawing geometry in the sketch.

Exercise 17-6 Complete the Exercise on the Student CD.

Creating Detail Views

Often part models contain complex geometry that is small in relation to the overall size of the part. These small features are typically difficult to see and dimension in a drawing. To overcome this problem, ***detail views*** are used to magnify the size of specific part features. For many parts, the use of detail views creates a cleaner, easier-to-understand drawing, and may be necessary for dimensioning purposes. See Figure 17-39.

Like projected views, detail views are generated from existing base views or dependent views, because model information is already stored in the drawing. The **Detail View** tool allows you to create detail views, which are projected and magnified from specified boundaries on an existing drawing view. Assuming at least one view referencing a model exists in the drawing; you can use the **Detail View** tool anytime during drawing development. To access the **Detail View** tool pick the **Detail View** button on the **Drawing Views Panel** bar; pick the **Detail View** button on the **Drawing Views Panel** toolbar; right-click on an existing parametric drawing view in the graphics window, and select **Detail** from the **Create View** cascading submenu; right-click on an existing parametric drawing view name in the browser, and select **Detail** from the **Create View** cascading submenu; or from the **Insert** pull-down menu, select the **Detail View** option of the **Model Views** cascading submenu.

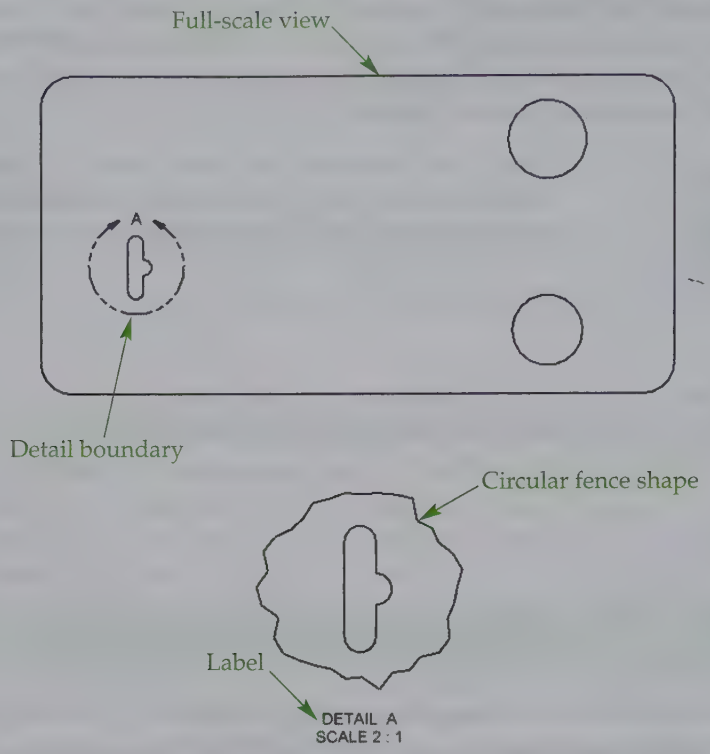
When you access the **Detail View** tool and a view is selected, the **Detail View** dialog box is displayed. See Figure 17-40.

NOTE

If you access the **Detail View** tool from the **Drawing Views Panel** bar, **Drawing Views Panel** toolbar, or the **Insert** pull-down menu, you must select the view from which you want to create the detail.

Use the **Label** area to modify the view label. You can edit the default name of the label in the **Label** edit box and pick the **Toggle Label Visibility** button if you want the label displayed in the view. The **Scale** area is used to define the scale of the view by

Figure 17-39.
An example of a detail view.



entering a scale in the **Scale** edit box, or select a scale from the flyout button. If you want the view scale to be displayed in the view, select the **Toggle Scale Visibility** button. The **Style** area allows you to specify the display of the view. You can choose to display the view with hidden lines by picking the **Hidden Lines** button, without hidden lines by picking the **Hidden Lines Removed** button, or as shaded by picking the **Shaded** button.

Once you specify the label, scale, and display options in the **Detail View** dialog box, the next step in creating a detail view is to define a boundary, or fence. The boundary consists of a phantom line circle, which represents the detail area being magnified. You can create a detail view with a circular or a rectangular fence, as shown in Figure 17-41. Placing a detail boundary is just like sketching a circle or a four-sided polygon in the Inventor sketch environment. To create a detail view with a circular fence, pick the **Circular Fence** button in the **Detail View** dialog box. Then in the drawing view, select the boundary center, followed by the boundary extents, or diameter of the circle. To create a detail view with a rectangular

Figure 17-40.
The **Detail View** dialog box.

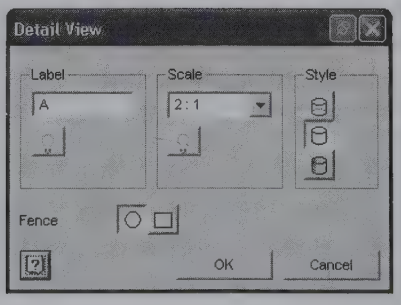
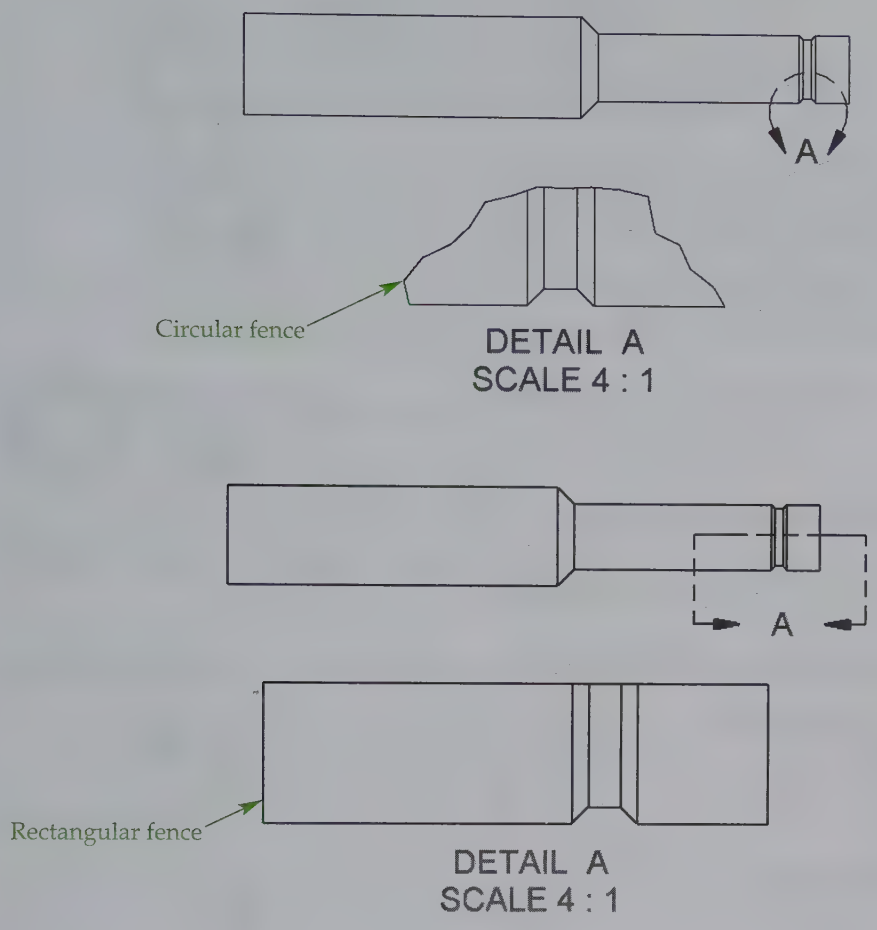


Figure 17-41.
A detail view can have a circular or rectangular fence.



fence, pick the **Rectangular Fence** button in the **Detail View** dialog box. Then select the boundary center, followed by the boundary extents, which is the edge of the rectangle. See Figure 17-42. Finally, generate the detail view by selecting the location where you want the view to be displayed, or by picking the **OK** button in the **Detail View** dialog box.

You will notice that the detail view boundary and labels are automatically created based on your specifications. Occasionally, as shown in Figure 17-43, the detail display is not acceptable. You can modify the following detail view components:

- **Boundaries.** Though you cannot directly edit the sketch of a boundary using sketch tools, you can change the size and location of the boundary, and the location of the boundary arrows and detail label. To change the location of the boundary by moving the existing center point, move your cursor over the boundary, or select the boundary until the boundary is highlighted and green dots appear. Then, pick the green dot in the center and drag the boundary to the desired location. Similarly, to increase or decrease the boundary size, move your cursor over the boundary or select the boundary until the boundary is highlighted and you see green dots. Then, pick one of the three green dots located around the boundary circumference and drag the boundary edge to the desired location. See Figure 17-44.

Figure 17-42.
Sketching a detail view with a boundary circle and a boundary rectangle.

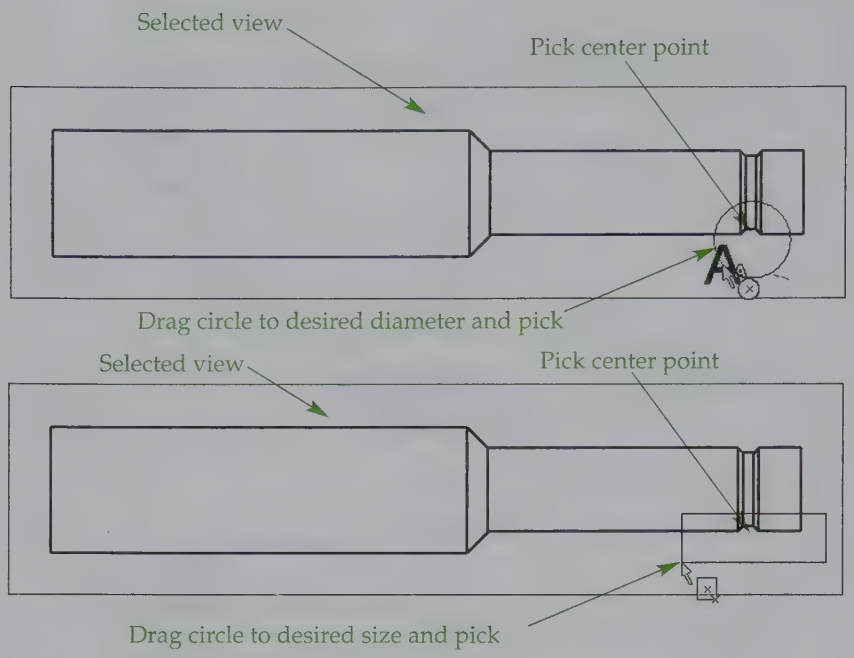
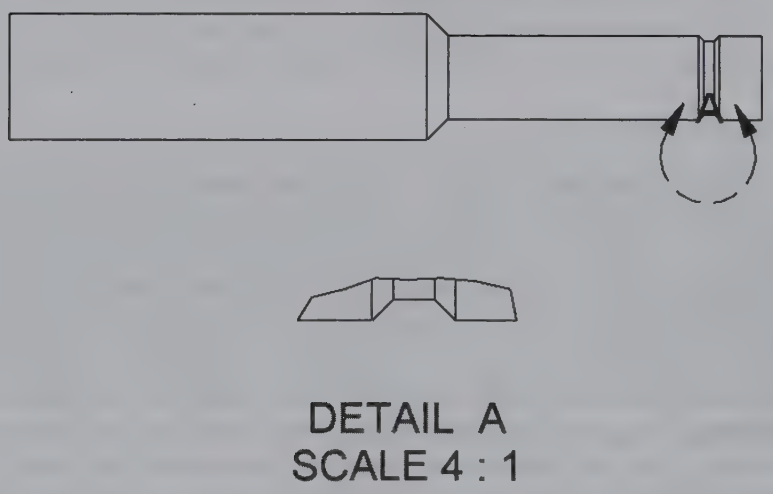


Figure 17-43.
An unsatisfactory detail display.



NOTE

When you adjust the size or location of the detail boundary, the detail view automatically updates to reflect the changes.

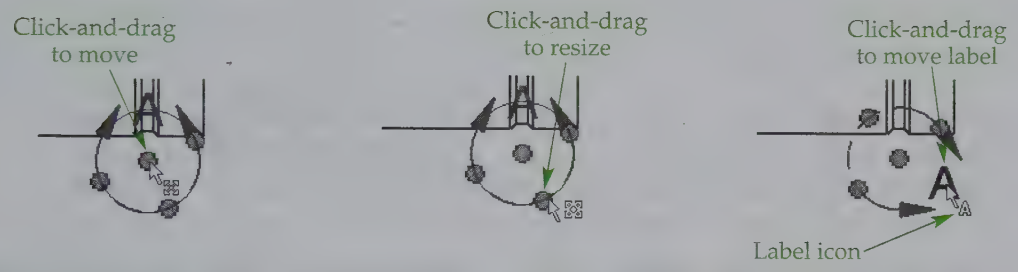
- **Boundary label.** To change the location of the boundary label, move your cursor over the boundary label until you see the **A** icon. Then select and drag the label to the desired location by rotating the boundary display. See Figure 17-44.
- **Label.** You can edit the default label parameters by right-clicking inside the detail view boundaries or the detail view name in the browser, and selecting the **Edit View...** menu option. However, you can modify and add information to the label text by right-clicking the label and selecting the **Edit View Label...** menu option to display the **Format Text** dialog box.

NOTE

Detail views are not bound by any alignment constraints and can be freely moved without affecting other views.

Exercise 17-7 Complete the Exercise on the Student CD.

Figure 17-44.
Detail boundaries can be moved, resized, and rotated to reposition the label.



Creating Conventional Breaks

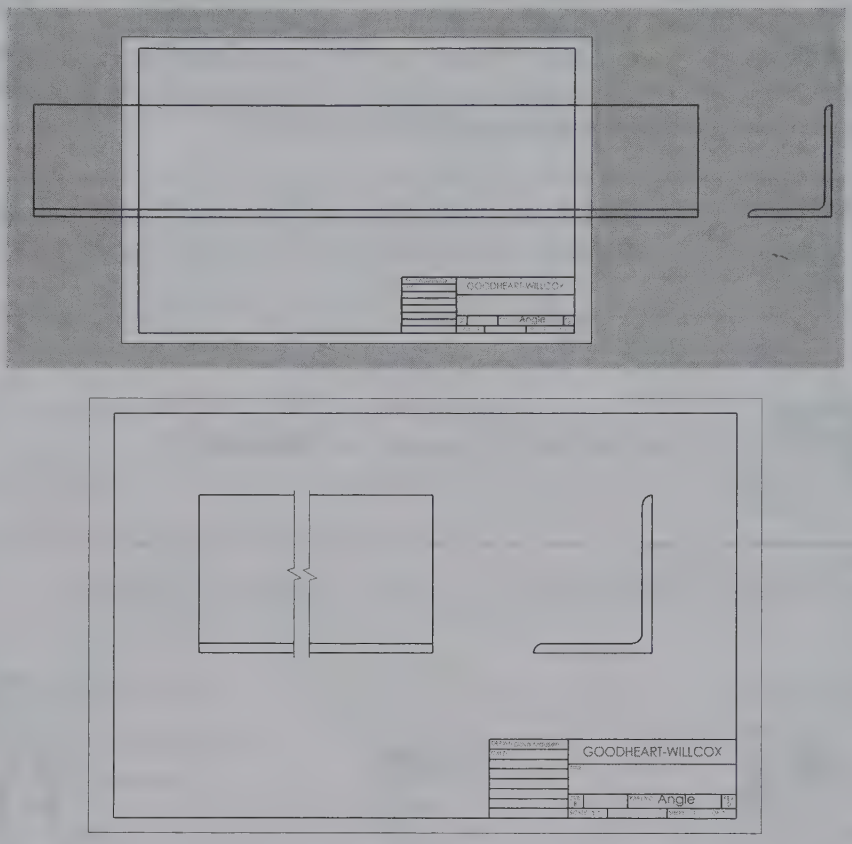
When you create a drawing of a long, constant shape part, the view may not fit on the desired paper size, or may look strange in reference to the rest of the drawing. To overcome this problem, a ***conventional break*** is used to shorten the view. For many long, consistently shaped parts, the use of a conventional break, or ***break***, is required to display views, or increase the view scale without increasing the paper size. See Figure 17-45.

NOTE

Breaking a drawing view does not change the actual parametric size, shape, or dimensions of the model.

Like projected views, conventional breaks are generated from existing base views or dependent views, because model information is already stored in the drawing. However, breaking a view modifies the existing view display, and does not actually project a new view.

Figure 17-45.
A long part that has a consistent shape can be shown in a broken view, so that its scale does not have to be reduced.



Conventional breaks are created using the **Broken View** tool, which can be used anytime during drawing development. To access the **Broken View** tool, pick the **Broken View** button on the **Drawing Views Panel** bar; pick the **Broken View** button on the **Drawing Views Panel** toolbar; right-click an existing parametric drawing view in the graphics window, and select **Broken** from the **Create View** cascading submenu; right-click on an existing parametric drawing view name in the browser, and select **Broken** from the **Create View** cascading submenu; or from the **Insert** pull-down menu, select the **Broken View** option of the **Model Views** cascading submenu.

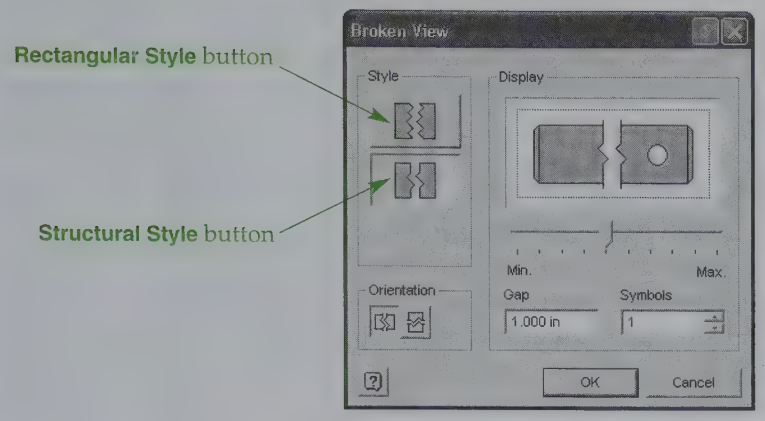
NOTE

If you access the **Broken View** tool from the **Drawing Views Panel** bar, **Drawing Views Panel** toolbar, or the **Insert** pull-down menu, you must select the view you want to break.

When you access the **Broken View** tool and a view is selected, the **Broken View** dialog box is displayed. See Figure 17-46. Use the following options in the **Broken View** dialog box to define the broken view display:

- **Style.** This area contains the **Rectangular Style** and **Structural Style** buttons. Pick the type of break style you want to use for your particular application. See Figure 17-47.
- **Orientation.** This area defines the orientation and direction of the break. Pick the **Horizontal Orientation** button when breaking a horizontal view, or pick the **Vertical Orientation** button when breaking a vertical view. See Figure 17-48.

Figure 17-46.
The **Broken View** dialog box.

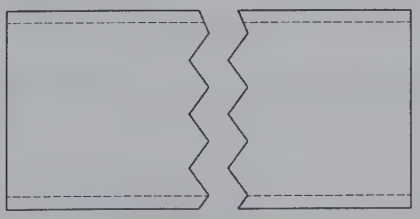


NOTE

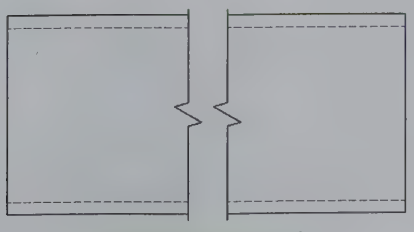
Typically, the correct orientation is automatically chosen, depending on the view you select.

- **Min./Max.** This slide bar allows you to adjust the size and number of break edges for rectangular style breaks, or the break symbol size when creating a structural style break. See Figure 17-49.
- **Gap.** Use this edit box to enter the gap or distance between break lines. The units specified in this edit box correspond to those in the current drawing.
- **Symbols.** This edit box is available only when creating a structural style conventional break and allows you to adjust the number of break symbols along the break lines.

Figure 17-47.
An example of a rectangular break style and a structural break style.



Rectangular break style



Structural break style

Figure 17-48.
An example of a horizontal orientation broken view and a vertical orientation broken view, each using a single break symbol.

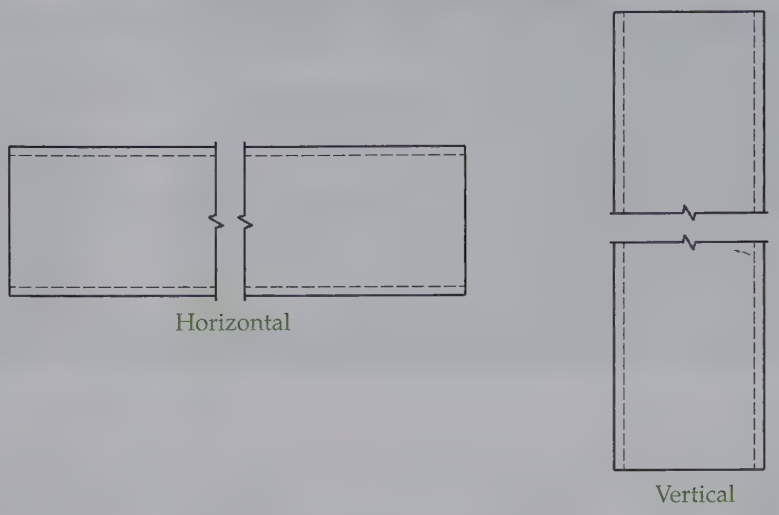
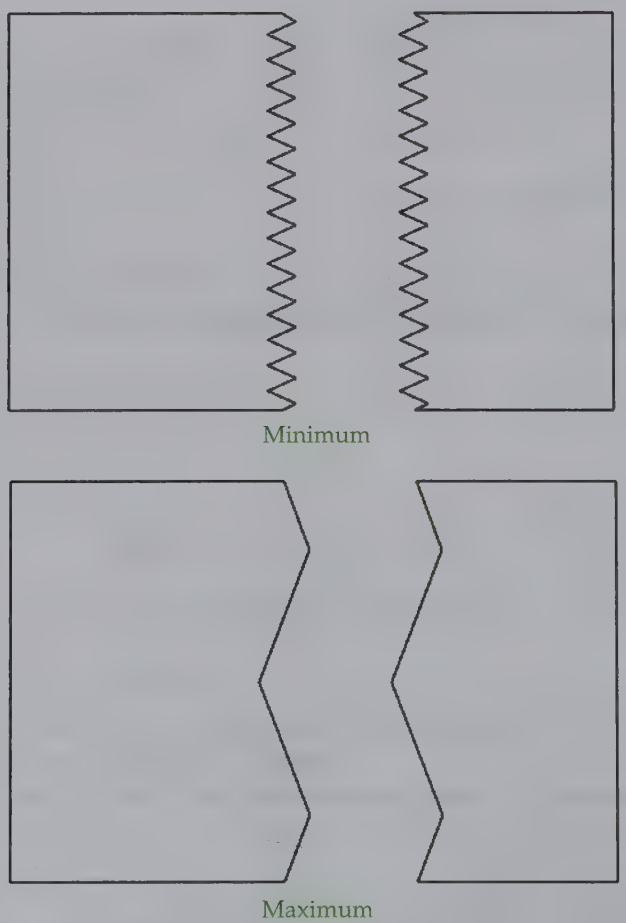


Figure 17-49.
A rectangular break style with a minimum edge size (large number of edges) and a rectangular break style with a maximum edge size.



Once you specify all the broken view display options in the **Broken View** dialog box, the next step in creating a conventional break is to define the break. The ***break*** is the portion of view that you want to remove and define as the broken length of the part. To specify the break, pick the location of the first break line and then select the location for the second break line. See **Figure 17-50.** The area between the first and second selections is removed by the break. Once you pick the location of the second break line, the broken view is generated.

To edit the break information you specified in the **Broken View** dialog box, right-click on the break lines and select the **Edit Break...** menu option. To change the location of the break lines without shortening or expanding the broken view, move your cursor over the break lines, or select the break lines until they are highlighted and you see a green dot. Then pick the green dot and drag the break lines to the desired location. If you want to shorten the break, select one of the break lines and drag it away from the center of the break lines. Similarly, to extend the break, drag one of the break lines toward and past the center of the break lines. See **Figure 17-51.**

NOTE

Inventor does not currently provide options for creating the ASME preferred appearance for cylindrical and tubular breaks.

Exercise 17-8 Complete the Exercise on the Student CD.

Figure 17-50.
Selecting break line locations.

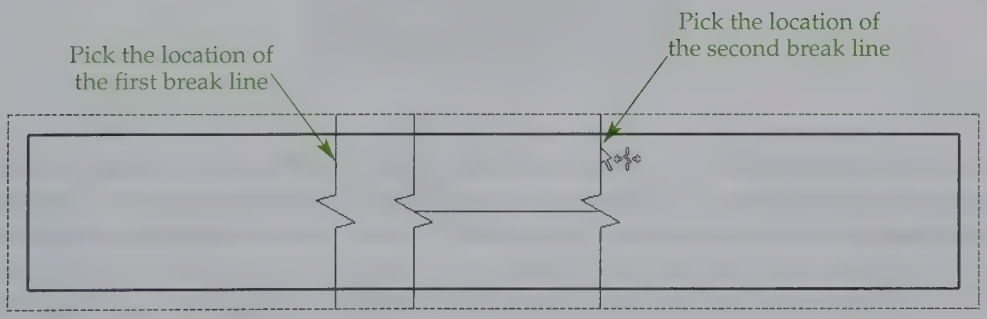
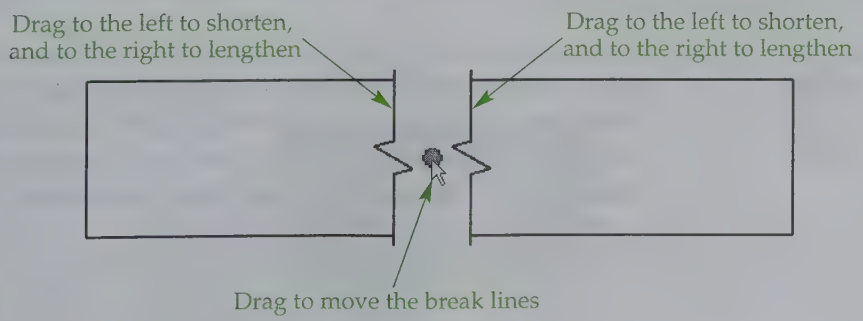


Figure 17-51.
Moving break lines and shortening or lengthening the broken view.

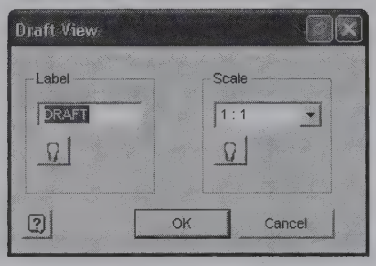


Sketching Views

The Inventor drawing environment is primarily used to create 2D drawings that reference 3D models. As a result, both the drawings and the models are parametrically controlled and change to reflect design intent. The ability to generate a 2D drawing quickly and easily from a 3D model is very powerful. However, if desired, you can produce drawing views using sketch tools. To sketch a drawing view, access the **Draft View** tool by picking the **Draft View** button on the **Drawing Views Panel** bar, picking the **Draft View** button on the **Drawing Views Panel** toolbar; or from the **Insert** pull-down menu, selecting the **Draft View...** option.

Once you access the **Draft View** tool, the **Draft View** dialog box is displayed. See Figure 17-52. Use the **Label** area to modify the view label. You can edit the default name of the label in the **Label** edit box and pick the **Toggle Label Visibility** button if you want the label displayed in the view. The **Scale** area is used to define the scale of the view by entering a scale in the **Scale** edit box, or select a scale from the flyout button. If you want the view scale to be displayed in the view, select the **Toggle Scale Visibility** button. The **Style** area allows you to specify the display of the view. You can choose to display the view with hidden lines by picking the **Hidden Lines** button, without hidden lines by picking the **Hidden Lines Removed** button, or as shaded by picking the **Shaded** button.

Figure 17-52.
The **Draft View** dialog box.



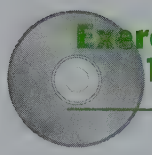
After you specify the view label and scale, pick the **OK** button to enter the sketch environment. Sketching a 2D draft view in a drawing file is the same as 2D sketching in any other Inventor file, such as a part. Refer to previous chapters and topics on specific drawing environment sketch tools for more information regarding sketches.

To edit the information you specified in the **Draft View** dialog box, right-click inside the draft view boundaries or the draft view name in the browser, and select the **Edit View...** menu option. You can also place a sketch on a drawing sheet by accessing the **Sketch** tool, available from the **Inventor Standard** toolbar, and using sketching tools to create geometry. If a view is selected, when you activate a sketch, the new sketch is associated with the selected view.

NOTE



When you finish drafting a draft view and exit the sketch environment, the dimensions you used to define the sketch are converted into drawing dimensions. The dimension style and specifications correspond to the current dimension settings.



Complete the Exercise on the Student CD.

Using Sheet Formats

Sheet formats are multi-element templates stored in the drawing file. When you use a sheet format, a new sheet is created, complete with views from a selected model, a border, and a title block, assuming these items are added to the sheet format. Sheet formats are located in the **Sheet Formats** folder of the **Drawing Resources** folder in the browser. See Figure 17-53.

By default, several sheet formats exist in a drawing file. To use one of these sheets, double-click the sheet format or right-click the sheet format and select the **New Sheet** menu option. Once the sheet format is selected, the **Select Component** dialog box appears. See Figure 17-54. If you have previously inserted a component, the model is available in the **Document Name** drop-down list. Otherwise, you need to pick the **Browse** button to display the **Open** dialog box and locate the model you want to insert. Then pick the **OK** button to create the drawing complete with a border, title block, and the specified views.

Figure 17-53.
The **Sheet Formats** folder contains the templates used in drawings.

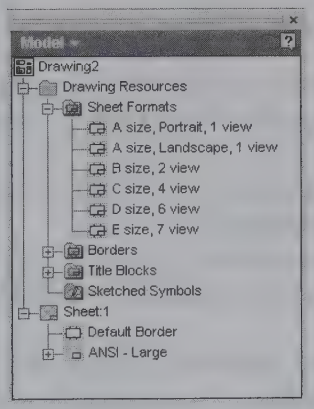
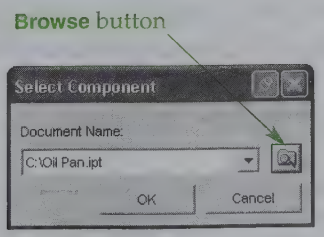


Figure 17-54.
Right-click a sheet format and select **New Sheet** to open the **Select Component** sheet format dialog box.

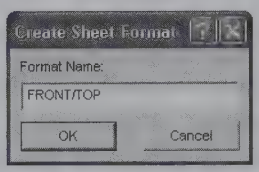


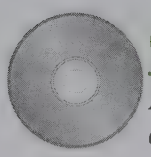
To create your own sheet format, generate a border and title block, and insert views from any model. The views you insert are not displayed when you create a new sheet and act only as a view reference. Right-click on the sheet in the **Browser** and select the **Create Sheet Format...** menu option, which displays the **Create Sheet Format** dialog box. See Figure 17-55. Enter the desired name in the **Format Name** edit box, and pick the **OK** button to create the sheet format.



Complete the Exercise on the Student CD.

Figure 17-55.
Enter the name of the new sheet in the **Format Name** edit box of the **Create Sheet Format** dialog box.





Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. What is a *monodetail drawing*?
2. Define *multidetail drawing.*
3. Regarding 2D geometry, what can you do once you insert a part model view?
4. What are 2D drawings most effectively built by referencing, and what is an alternative?
5. Identify at least two of the options used to create drawing views in Inventor.
6. Give the function of the **Base View** tool.
7. Briefly describe a *perspective drawing.*
8. Explain why it is easy to quickly generate projected views; and name the tool used.
9. What are *auxiliary views* and when are they needed?
10. Describe *partial auxiliary views.*
11. Provide another name for *sectional views* and give a definition.
12. Name the line that represents the cutting plane of the section, and identify where it is located.
13. What do section lines represent?
14. Give another name for *broken-out section views*, and explain how they are different from other types of section views.
15. Identify the function of *detail views.*
16. When is a *conventional break* used, and what is another name?
17. Why is the ability to create a 2D drawing from a 3D model an important feature of Inventor?
18. Name the tool used to sketch a drawing view.
19. Define *sheet formats.*
20. What is included when you use a sheet format, assuming these items were added to the sheet format?

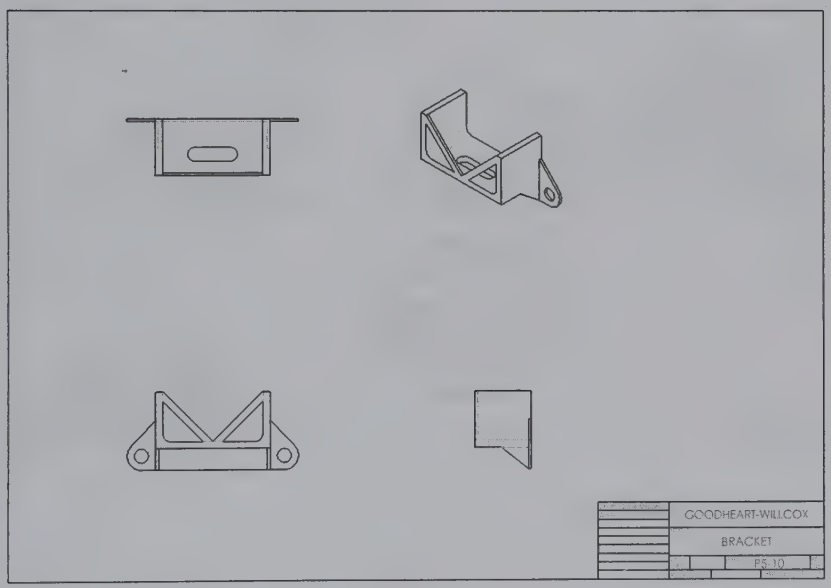
Problems

Instructions:

- Begin a new drawing file for each of the projects.
- Modify the sheet size and layout as needed.
- Use the information provided to create the drawings shown.
- Define the drawing properties in the **Properties** dialog box, so the properties are displayed in the title block.
- Do not add any dimensions or annotations to the drawings.

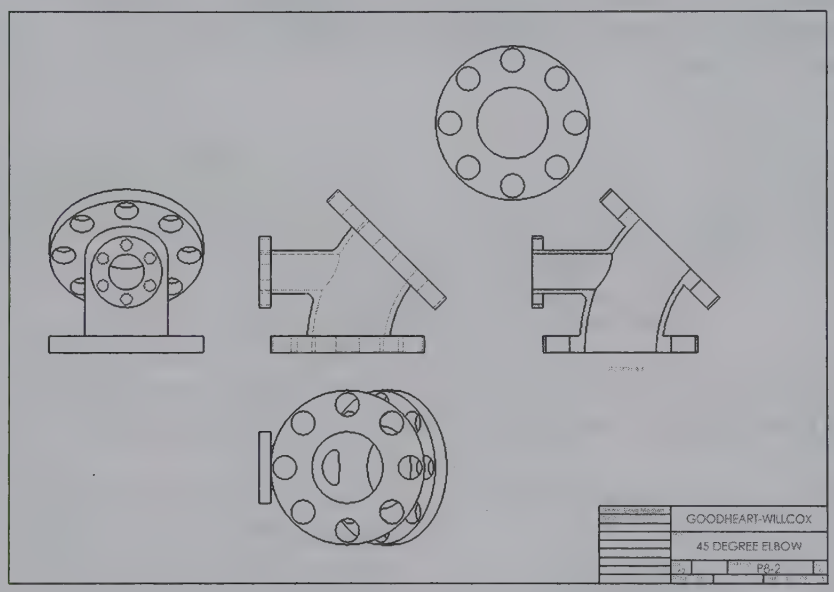
1. **Name:** Bracket
   Units: Metric
   Template: DRAWING-mm.idw
   Sheet Size: A2
   Drawing View Scale (unless otherwise noted): 1:1
   Base View File: P5-10.ipt
   Specific Instructions:
   - Place the base view first using the **Base View** tool.
   - Project the three additional views using the **Project View** tool.

   The final drawing should look similar to the drawing shown.
   Save As: P17-1.idw



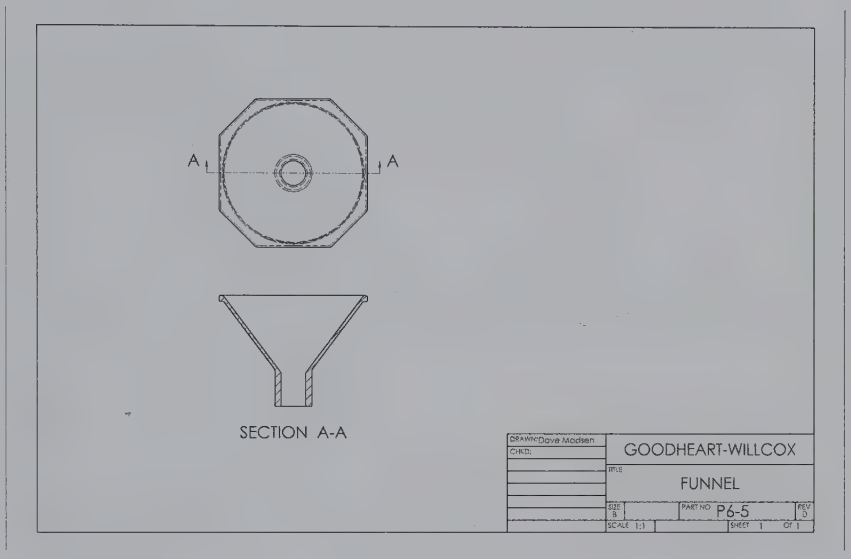
2. **Name:** 45° Elbow
Units: Metric
Template: DRAWING-mm.idw
Sheet Size: A2
Drawing View Scale (unless otherwise noted): 1:1
Base View File: P8-2.ipt
Specific Instructions:
   - Place the front view first using the **Base View** tool.
   - Project the side and bottom views using the **Project View** tool.
   - Edit the side and bottom views by removing the **Style from Base** view constraint and selecting the **Hidden Lines Removed** button.
   - Create the auxiliary view using the **Auxiliary View** tool.
   - Create the section view using the **Section View** tool.
   - Align the section view with the front view.

   The final drawing should look similar to the drawing shown.
   Save As: P17-2.idw



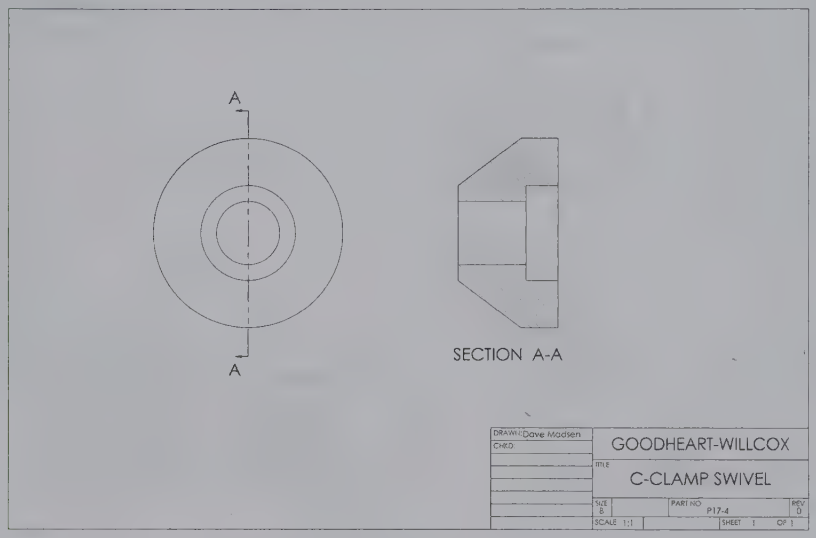
3. **Name:** Funnel
Units: Inch
Template: DRAWING-in.idw
Sheet Size: B
Drawing View Scale (unless otherwise noted): 1:1
Base View File: P6-5.ipt
Specific Instructions:
- Place the top view first using the **Base View** tool.
- Create the section view using the **Section View** tool.

The final drawing should look similar to the drawing shown.
Save As: P17-3.idw



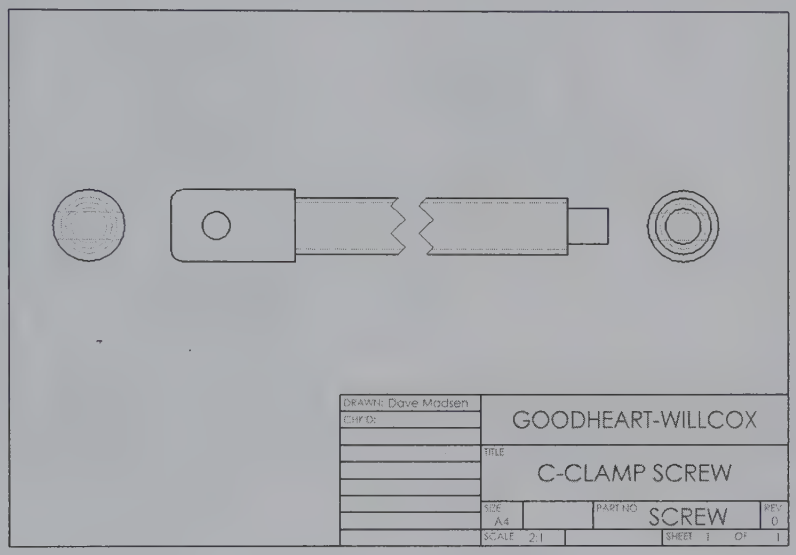
4. **Name:** C-Clamp Swivel
Units: Metric
Template: DRAWING-mm.idw
Sheet Size: A4
Drawing View Scale (unless otherwise noted): 2:1
Base View File: P6-2.ipt
Specific Instructions:
- Place the top view first using the **Base View** tool.
- Create the section view using the **Section View** tool.

The final drawing should look similar to the drawing shown.
Save As: P17-4.idw



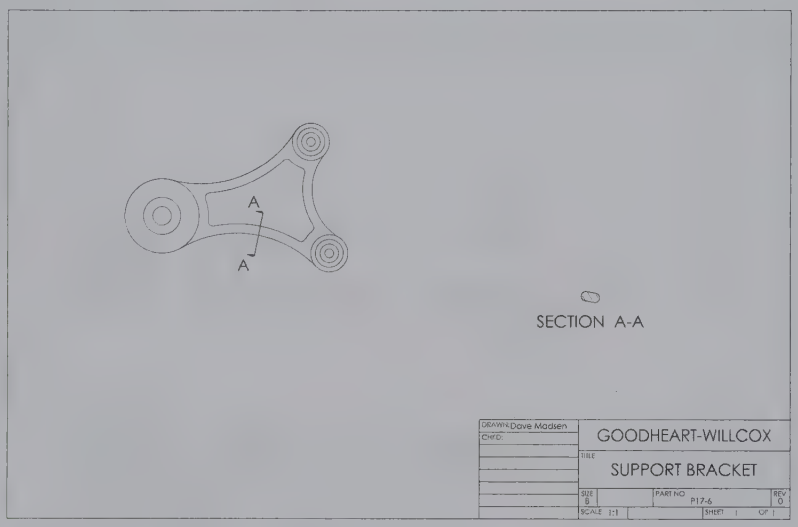
5. **Name:** C-Clamp Screw
Units: Metric
Template: DRAWING-mm.idw
Sheet Size: A4
Drawing View Scale (unless otherwise noted): 2:1
Base View File: P8-6.ipt
Specific Instructions:
   - Place the side view first using the **Base View** tool.
   - Break the side view using the **Broken View** tool.
   - Create the side views using the **Project View** tool.

   The final drawing should look similar to the drawing shown.
Save As: P17-5.idw



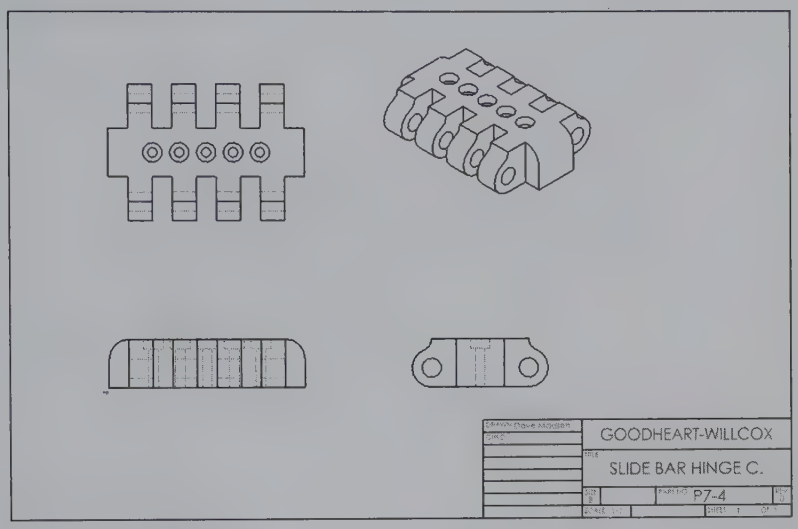
6. **Name:** Support Bracket
Units: Inch
Template: DRAWING-in.idw
Sheet Size: B
Drawing View Scale (unless otherwise noted): 3:1
Base View File: P6-4.ipt
Specific Instructions:
- Place the top view first using the **Base View** tool.
- Create the section view using the **Section View** tool.

The final drawing should look similar to the drawing shown.
Save As: P17-6.idw



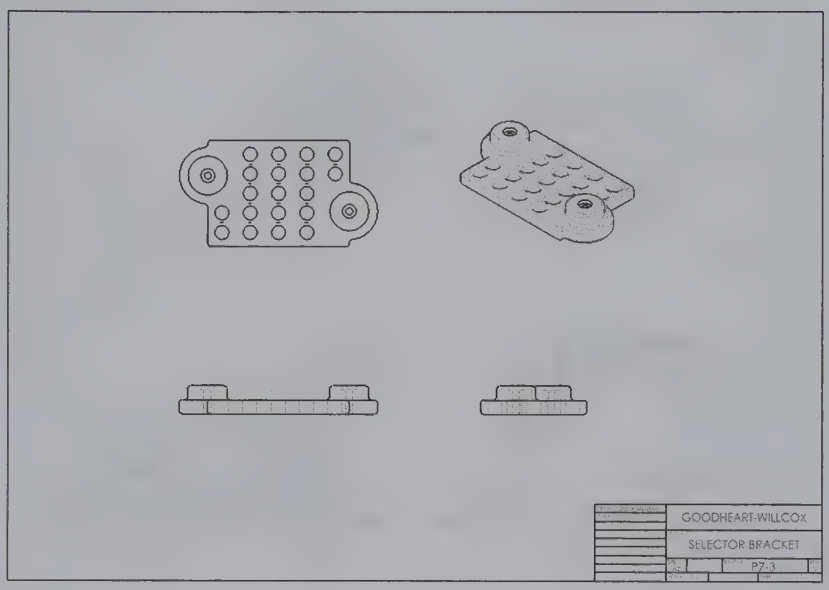
7. **Name:** Slide Bar Hinge Connector
Units: Inch
Template: DRAWING-in.idw
Sheet Size: B
Drawing View Scale (unless otherwise noted): 1:1
Base View File: P7-4.ipt
Specific Instructions:
- Place the front view first using the **Base View** tool.
- Create the additional views using the **Project View** tool.

The final drawing should look similar to the drawing shown.
Save As: P17-7.idw



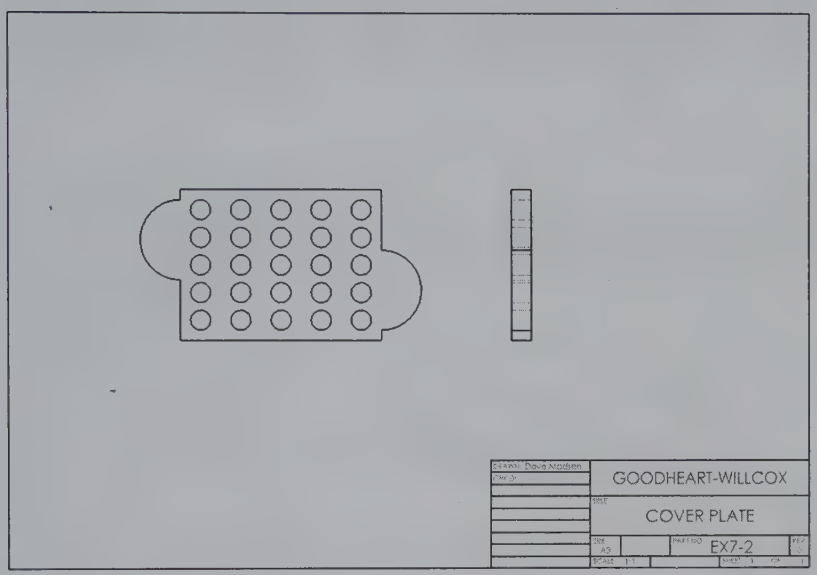
8. **Name:** Selector Bracket
Units: Metric
Template: DRAWING-mm.idw
Sheet Size: A2
Drawing View Scale (unless otherwise noted): 1:1
Base View File: P7-3.ipt
Specific Instructions:
- Place the front view first using the **Drawing Sketch Panel** tool.
- Create the additional views using the **Project View** tool.

The final drawing should look similar to the drawing shown.
Save As: P17-8.idw



9. **Name:** Cover Plate
Units: Metric
Template: DRAWING-mm.idw
Sheet Size: A4
Drawing View Scale (unless otherwise noted): 1:1
Base View File: EX7-2.ipt
Specific Instructions:
- Place the front view first using the **Base View** tool.
- Create the additional view using the **Project View** tool.

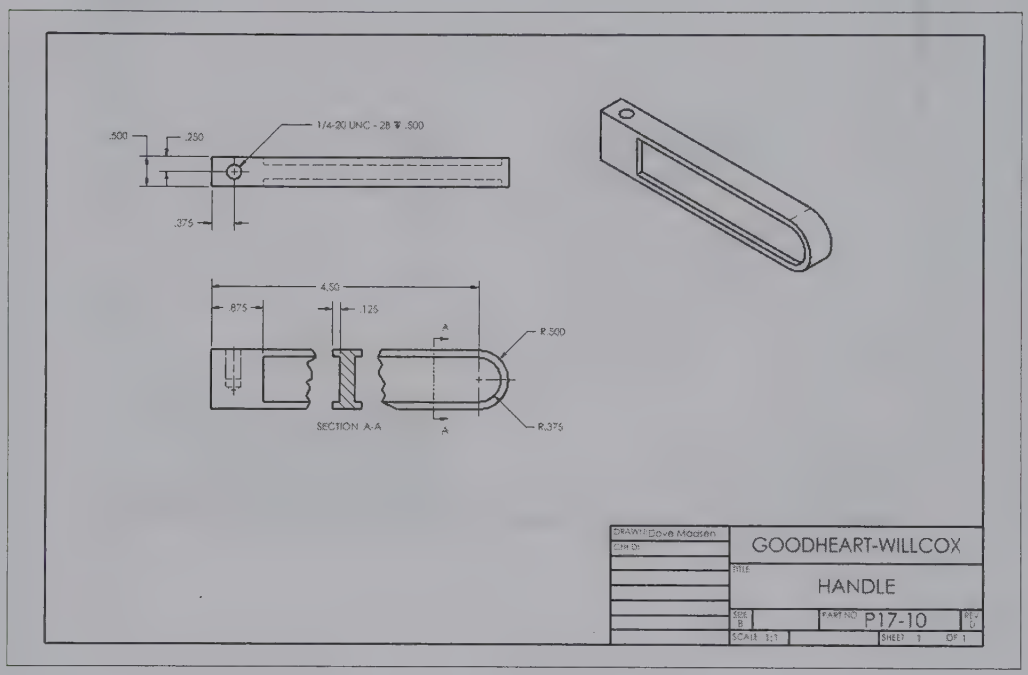
The final drawing should look similar to the drawing shown.
Save As: P17-9.idw

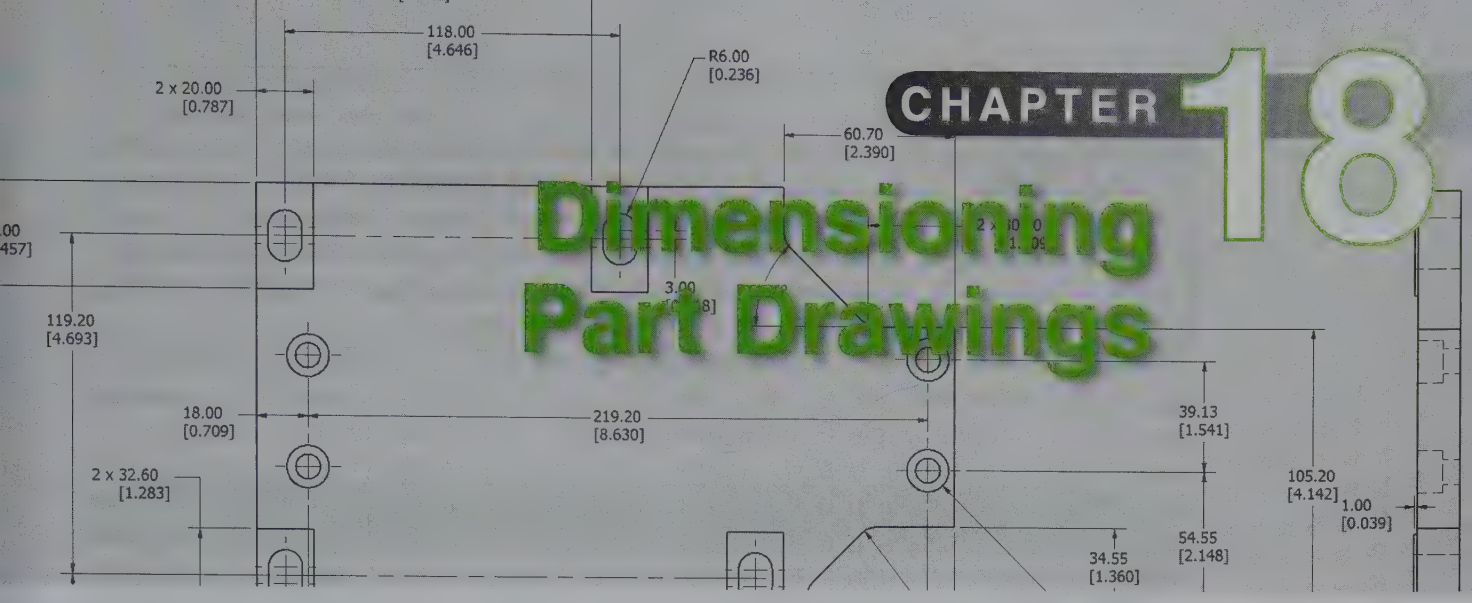


10. **Name:** Handle
Units: Inch
Template: DRAWING-in.idw
Sheet Size: B
Drawing View Scale (unless otherwise noted): 1:1
Base View File: P17-10.ipt
Specific Instructions:

- Use the drawing shown to generate a part model.
- Save the part model as P17-10.ipt.
- Open a new drawing file, and place the drawing views shown. Do not dimension the drawing.

The final drawing should look similar to the drawing shown.
Save As: P17-10.idw





CHAPTER 18

Dimensioning Part Drawings

Learning Objectives

After completing this chapter, you will be able to do the following:

- Work with, and differentiate between, model and drawing dimensions.
- Add centerlines and center marks.
- Use the general dimension tool.
- Create datum dimensions.
- Add general and local notes.
- Use ordinate and tabular dimension tools.
- Add hole and thread notes.
- Place surface texture symbols.
- Use geometric dimensioning and tolerancing tools.
- Create custom drawing symbols.
- Add a revision table and revision tags.
- Modify dimensions.

Once you place drawing views, the next step to create a drawing is to add dimensions to view geometry. ***Dimensions*** define the size and shape of object features so that parts can be manufactured. Dimensions, along with other notes and text, also specify the location and characteristics of geometry and surface texture.

Inventor provides multiple tools that allow you to create a wide variety of drawing dimensions, notes, and text information. These tools are referred to as ***drawing annotation tools***. Throughout this chapter, you will explore specific tools and options that allow you to fully dimension a part drawing.

NOTE



The **Welding Symbol**, **Balloon**, **Auto Balloon**, **Parts List**, **Caterpillar**, and **End Treatment** drawing annotation tools correspond to assembly and weldment drawings. These tools will be described later in this book.

PROFESSIONAL TIP



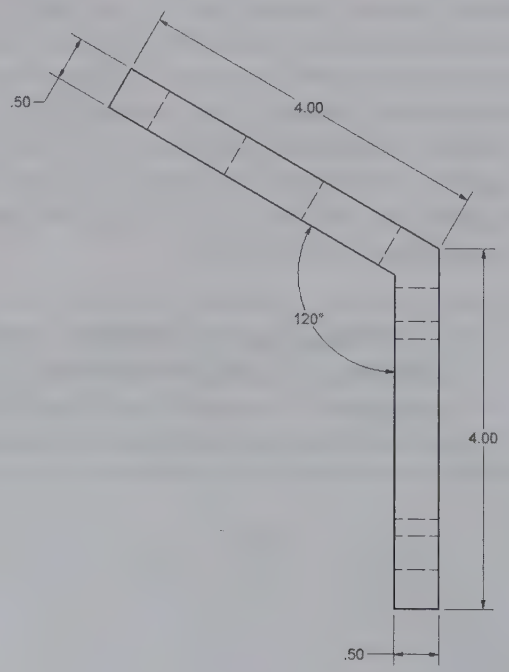
Dimension and annotation preferences are established primarily in the **Styles and Standards Editor**. Unless you plan to use default style and standard settings, you should completely understand drawing styles and standards in order to make a drawing look exactly the way you want it to look, and capture design intent.

Working with Model Dimensions

Most part drawings are generated from existing 3D parametric solid models. As a result, Inventor drawing files contain model and drawing dimensions. ***Model dimensions*** are specified when you add dimensions and constraints to a model sketch, or define the size of an object, such as an extrusion depth. Model dimensions can be displayed automatically when you insert a model into a drawing. Model dimensions are parametric and, when edited, change the physical size and shape of the corresponding model. Similarly, if you modify model dimension while in a model file, the model dimension and the size and shape of the object in the corresponding drawing also change. See **Figure 18-1**.

Only model dimensions that are planar to the drawing view can be shown in a view. For example, if the thickness of a part is shown in the drawing view, the model dimension that defines the part thickness can be shown. There are three primary options for displaying model dimensions. One technique requires that you access the **Drawing Tab** of the **Options** dialog box before you create a base view, and choose the **Retrieve all model dimensions on view placement** check box. Now when you insert a drawing view,

Figure 18-1.
An example of model dimensions in a drawing view. Often it may be difficult to differentiate between model and drawing dimensions.



the model dimensions should automatically be displayed. A second option is to pick the **All Model Dimensions** check box in the **Options** tab of the **Drawing View** dialog box when inserting or editing a base view. The third option can be used to display model dimensions once a view has been placed, using the **Retrieve Dimensions** dialog box. Access the **Retrieve Dimensions** dialog box by right-clicking on a drawing view in the graphics window and selecting the **Retrieve Dimensions...** pop-up menu option; right-click on a drawing name in the browser and select the **Retrieve Dimensions...** pop-up menu option; pick the **Retrieve Dimensions** on the **Drawing Annotation Panel** bar; or pick the **Retrieve Dimensions** on the **Drawing Annotation Panel** toolbar. Once you access the **Retrieve Dimensions** dialog box, shown in Figure 18-2, pick the **Select View** button, if it is not already selected, and choose a view from which to reference a model and display model dimensions. This step is not required if you accessed the tool by right-clicking on a view. Next, if you want to retrieve dimensions for specific model features, pick the **Select Features** radio button, followed by choosing model features in the graphics window. Or, if you want to retrieve dimensions for entire parts, pick the **Select Parts** radio button, followed by selecting parts in the graphics window or from the browser. When you choose features or parts, the available model dimensions are automatically shown. The next step is to pick the **Select Dimensions** button, followed by choosing the model dimensions that you want to display in the drawing. To continue using the **Retrieve Dimensions** dialog box, pick the **Apply** button to generate the selected dimensions, and repeat the steps previously described. When you are finished retrieving model dimensions, pick the **OK** button to exit the **Retrieve Dimensions** dialog box.

Model dimensions are often placed directly over view geometry or in unacceptable positions. You can move model dimensions by selecting the dimension text and dragging the entire dimension to the desired location. You can never actually remove a model dimension from a drawing, unless you delete the parameter from the model in the part file. However, you can hide individual model dimensions by right-clicking on the dimension you want to hide and selecting the **Delete** menu option. You can also hide all model dimensions in a view by right-clicking inside a view or the view name in the browser and deselecting the **Model Dimensions** option of the **Annotation Visibility** cascading submenu.

If you want to change the size or shape of a model while working in the drawing environment, right-click the model dimension of the geometry you want to change, and select the **Edit Model Dimension...** menu option. When you pick the **Edit Model Dimension...** menu option, the **Edit Dimension** dialog box for the selected parameter is displayed. See Figure 18-3. Enter or select the new parameter value in the edit box and pick the **OK** button. It may take some time for the changes to occur, because the geometry of the model is also modified to reflect the new value.

Figure 18-2.
The **Retrieve Dimensions** dialog box.

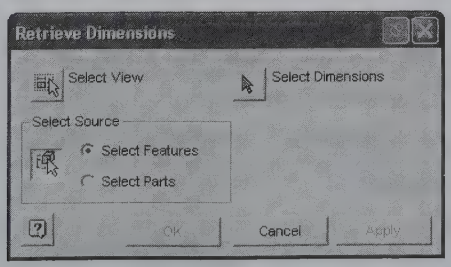


Figure 18-3.
The **Edit Dimension** dialog box.





Complete the Exercise on the Student CD.

Drawing Dimensions

Often model dimensions do not fully document design intent, or are not appropriate according to correct drafting practices. As a result, ***drawing dimensions*** are used to place additional, or correct dimensions on drawing view geometry to document the design for manufacturing purposes. Unlike model dimensions, drawing dimensions do not control any model parameters. Still, when you edit a model while in a model file, the drawing dimensions in the drawing file change to reflect the new values. However, you cannot change the size or shape of a model by manipulating drawing dimensions.

Drawing annotation tools are used to create a wide variety of drawing dimensions and notes; including linear, angular, radial, diameter, and ordinate dimensions, hole and thread notes, and surface texture symbols, just to name a few. These tools are accessed from the **Drawing Annotation Panel** toolbar, from the **Drawing Annotation Panel** bar, or from specific drawing annotation toolbars. Once a drawing view is created, you can enter the drawing annotation work environment by right-clicking on the panel bar and selecting the **Drawing Annotation Panel** menu option.

Adding Centerlines and Center Marks

Drawings that contain circles, arcs, other circular geometry, or symmetrical features require center marks or centerlines in order to be fully dimensioned and describe the center and location of the features. Inventor provides you with two primary options for placing drawing centerlines and center marks. One option is to use automated centerlines, which automatically places center marks and centerlines on the desired drawing geometry. The other option is to use specific tools that allow you to place center marks on single circles and arcs, on linear and circular groups of circles and arcs, and centerline bisectors. See Figure 18-4. Use the following tools and techniques to add centerlines and center marks to your drawings:

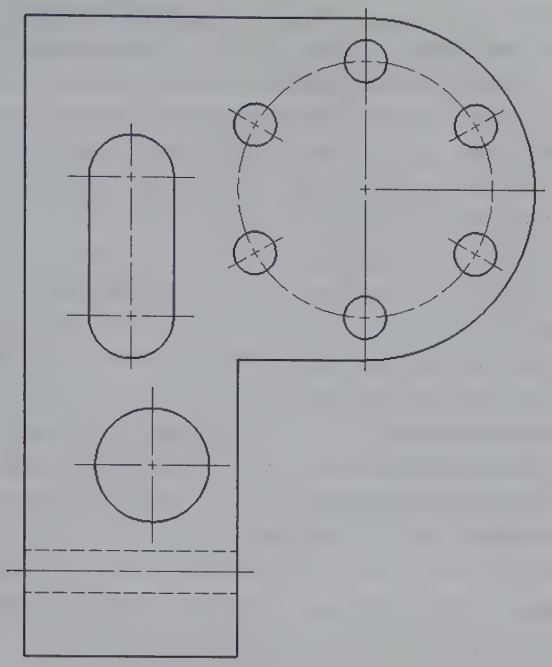
NOTE



The **Center Mark**, **Centerline**, **Centerline Bisector**, and **Centered Pattern** tools are all located in a flyout button in the **Drawing Annotation Panel** toolbar and panel bar.

- **Automated Centerlines.** Automated centerlines are defined using the **Automated Centerlines** dialog box. To apply automated centerlines to the entire document, access the **Automated Centerlines** dialog box from the **Document Settings** dialog box by picking the **Documents Settings...** option of the **Tools** pull-down menu.

Figure 18-4.
Examples of the types of center marks and centerlines you can create.



Setting automated centerlines using the **Document Settings** dialog box was previously described, because you may want to set automated centerlines in your drawing templates. Another option for automatically adding centerlines and center marks is to apply automated centerlines to a specific view using the **Automated Centerlines** dialog box. To use this technique, right-click on an existing drawing view and select the **Automated Centerlines...** pop-up menu option. When you access the **Automated Centerlines** dialog box from a single view, the settings you specify in the dialog box affect only the selected view.

- **Center Mark.** This tool is used to add center marks to a variety of circular geometric shapes, such as circles, ellipses, and arcs. See Figure 18-5. To access the **Center Mark** tool, pick the **Center Mark** button on the **Drawing Annotation Panel** bar; pick the **Center Mark** button on the **Drawing Annotation Panel** toolbar; or pick the **Center Mark** button on the **Drawing Center Marks** toolbar. Use the **Center Mark** tool by picking the geometry or the center of the geometry on which you want to place a center mark. Continue placing center marks as required, and when finished, press the [Esc] key on your keyboard, or right-click and select the **Done** menu option.

NOTE



To create a center mark without extension lines, right-click before you place the center mark, and deselect the **Extension lines** pop-up menu option.

- **Centerline.** This tool is used to add center marks to two or more round objects with connected centerlines. See Figure 18-6. To access the **Centerline** tool, pick the **Centerline** button on the **Drawing Annotation Panel** bar; pick the **Centerline** button on the **Drawing Annotation Panel** toolbar; pick the **Centerline** button on the **Drawing Center Marks** toolbar; or while using either the **Centered Pattern** or **Centerline Bisector** tools, right-click and select **Free Points** from the **Type**

Figure 18-5.
Example of various center marks created using the **Center Mark** tool.

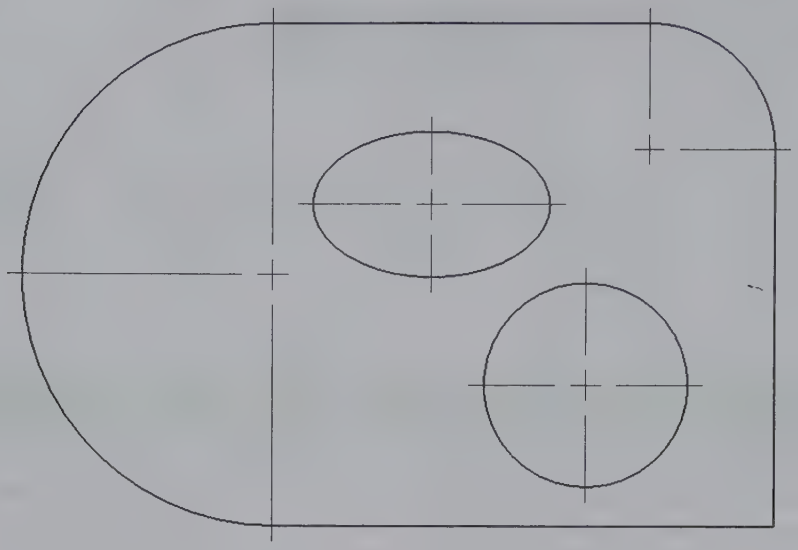
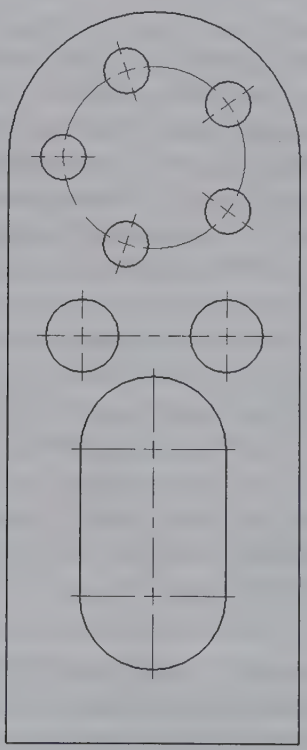


Figure 18-6.
Example of various center marks created using the **Centerline** tool.



cascading submenu. Use the **Centerline** tool by first picking the geometry or the center of the geometry on which you want to place the first center mark. Next, continue picking additional objects as required. Then, generate the center marks by right-clicking and selecting the **Create** menu option. Continue placing center marks as needed. When finished, press the [Esc] key on your keyboard or right-click and select the **Done** menu option.

- **Centered Pattern.** This tool is similar to the **Centerline** tool but is primarily used to add center marks to multiple circular objects and a bolt circle when applying polar coordinate dimensions. See Figure 18-7. To access the **Centered Pattern** tool, pick the **Centered Pattern** button on the **Drawing Annotation Panel** bar; pick the **Centered Pattern** button on the **Drawing Annotation Panel** toolbar; pick the **Centered Pattern** button on the **Drawing Center Marks** toolbar; or while using either the **Centerline** or **Centerline Bisector** tools, right-click and select **Centered Pattern** from the **Type** cascading submenu. Use the **Centered Pattern** tool by first selecting the outside diameter or radius of the part. Then, in a clockwise or counterclockwise fashion, select the objects in the pattern. See Figure 18-7. To generate the center marks, right-click and select the **Create** menu option. Continue placing center marks on other objects as needed. When finished, press the [Esc] key on your keyboard or right-click and select the **Done** menu option.
- **Centerline Bisector.** This tool allows you to place a centerline that bisects two symmetrical objects, such as a hidden or section view of a hole. See Figure 18-8. To access the **Centerline Bisector** tool, pick the **Centerline Bisector** button on the **Drawing Annotation Panel** bar; pick the **Centerline Bisector** button on the **Drawing Annotation Panel** toolbar; pick the **Centerline Bisector** button on the **Drawing Center Marks** toolbar; or while using either the **Centerline** or **Centered Pattern** tools, right-click and select **Bisector** from the **Type** cascading submenu. Use the **Centerline Bisector** tool by first selecting the first side of the symmetrical object, followed by the opposite side. See Figure 18-8. Then, generate the centerline by right-clicking and selecting the **Create** menu option. Continue placing centerline bisectors on other objects as needed. When finished, press the [Esc] key on your keyboard or right-click and select the **Done** menu option.

Figure 18-7.
An example of the center marks created using the **Centered Pattern** tool.

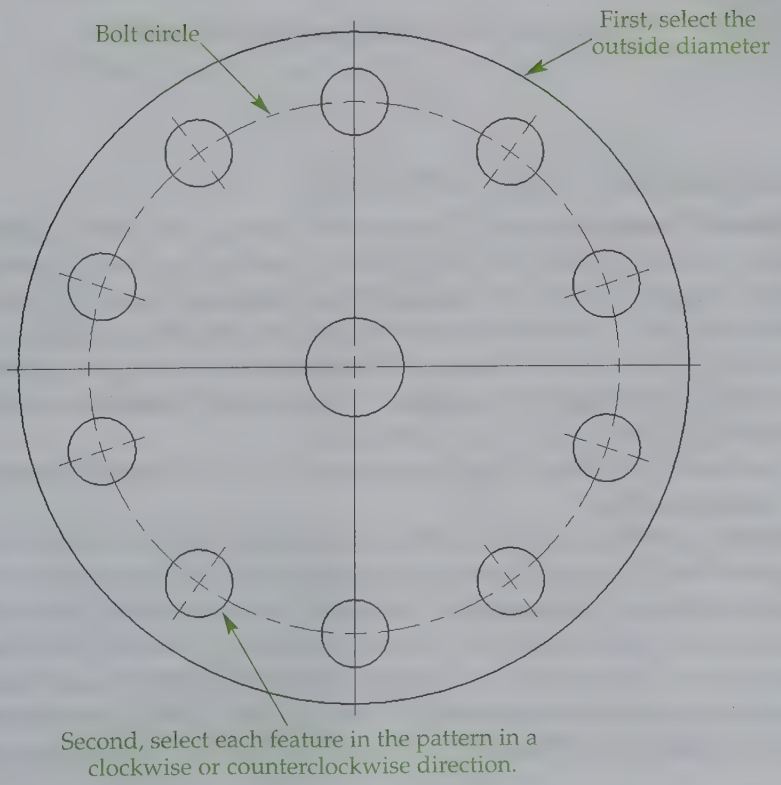
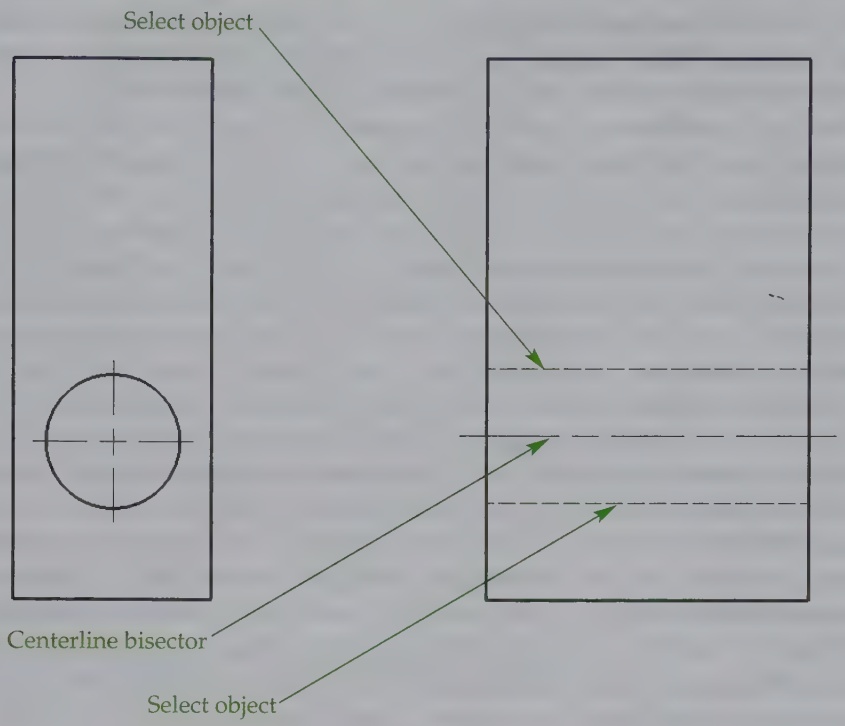


Figure 18-8.
An example of a centerline bisector created using the **Centerline Bisector** tool.



Exercise 18-2 Complete the Exercise on the Student CD.

Exercise 18-3 Complete the Exercise on the Student CD.

Using the General Dimension Tool

The **General Dimension** tool allows you to create a number of different drawing dimensions, including linear, aligned, radial, diameter, and angular dimensions. See Figure 18-9. General dimensions are placed on a drawing using slightly different techniques, depending on the type of geometry you are dimensioning. Access the **General Dimension** tool by picking the **General Dimension** button on the **Drawing Annotation Panel** bar; picking the **General Dimension** button on the **Drawing Annotation Panel** toolbar; or pressing the [D] key on your keyboard.

You can create the following dimensions with the **General Dimension** tool using the techniques described:

- **Linear.** *Linear* dimensions define and locate horizontal and vertical geometry. See Figure 18-10. To create a linear dimension, select the object you want to dimension, the endpoints of the objects, or an object and an endpoint. The selection method depends on the dimension you want to create. Then, drag the dimension line to the desired location and pick the position. See Figure 18-11. The dimension will become dotted when you drag the dimension over the location designated in the **Spacing** and **Part Offset** edit boxes of the **Dimension Styles** area, **Display** tab of

Figure 18-9.
Examples of dimensions created using the **General Dimension** tool.

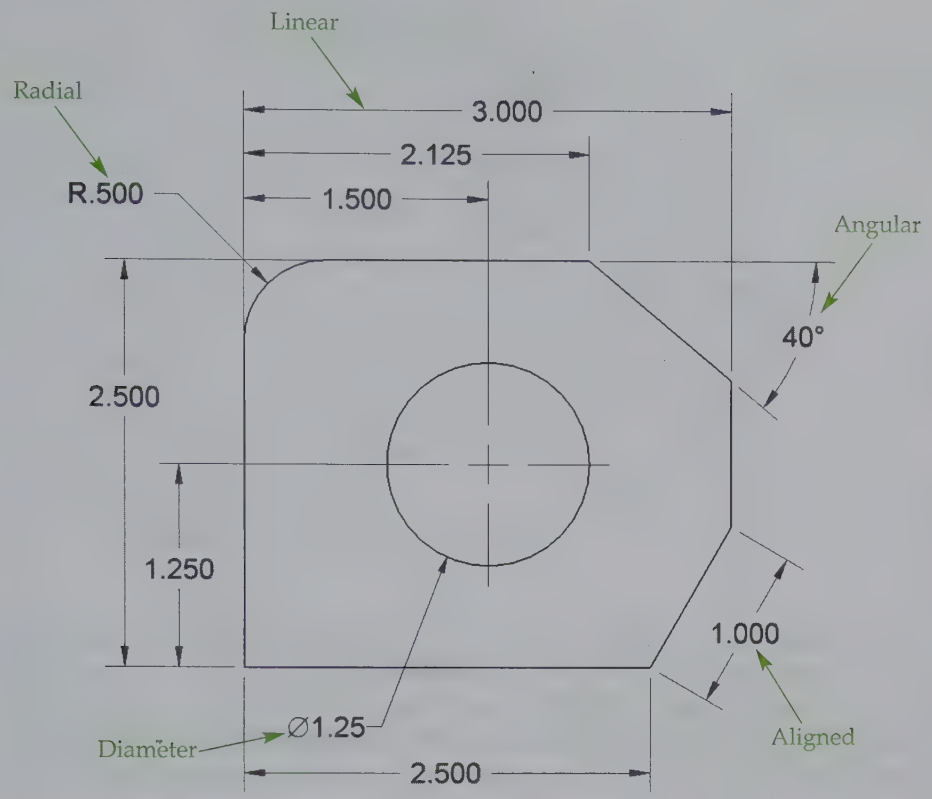


Figure 18-10.
Selecting objects to apply general dimensions.

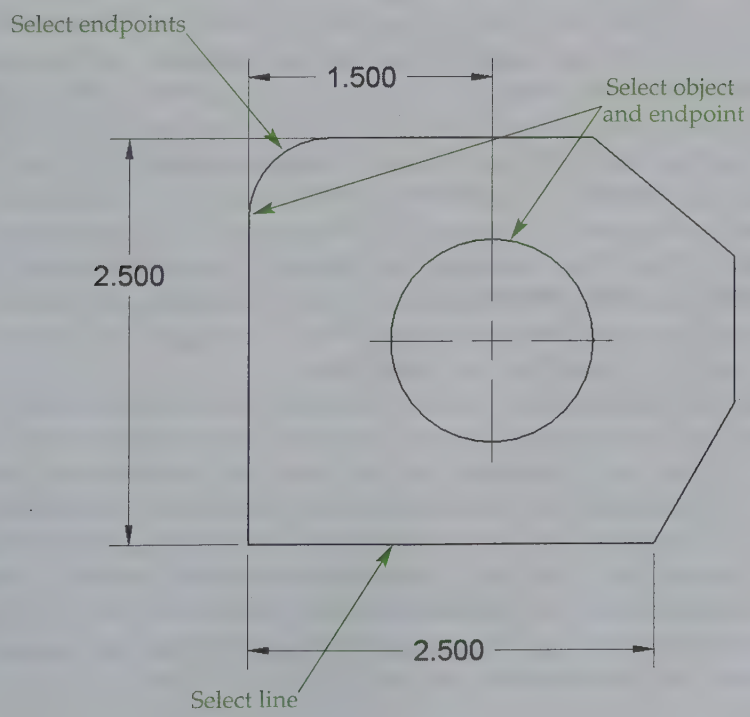
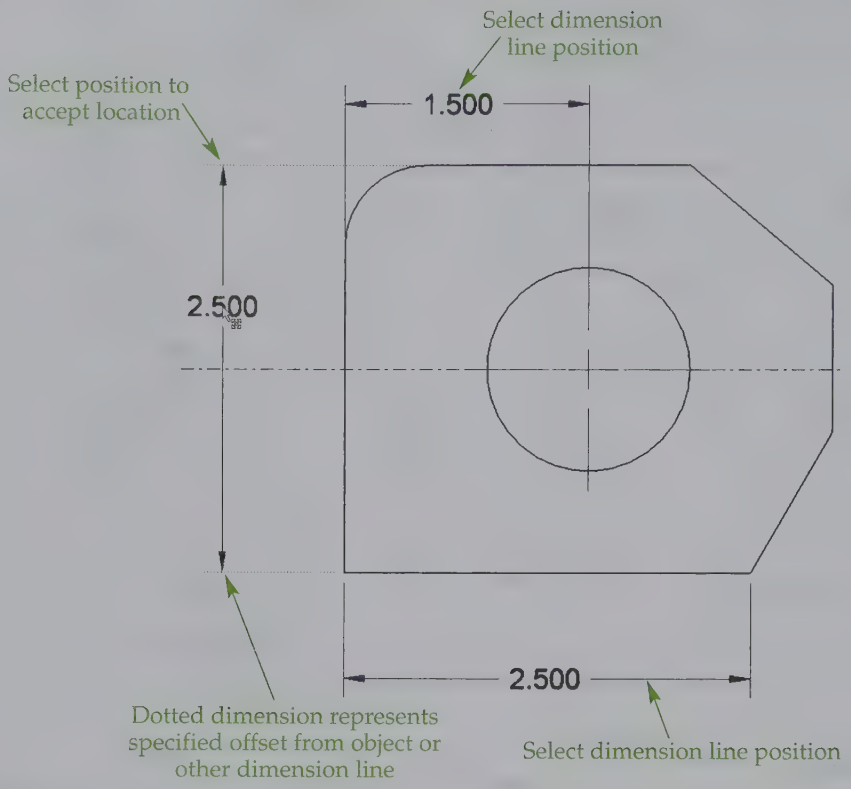


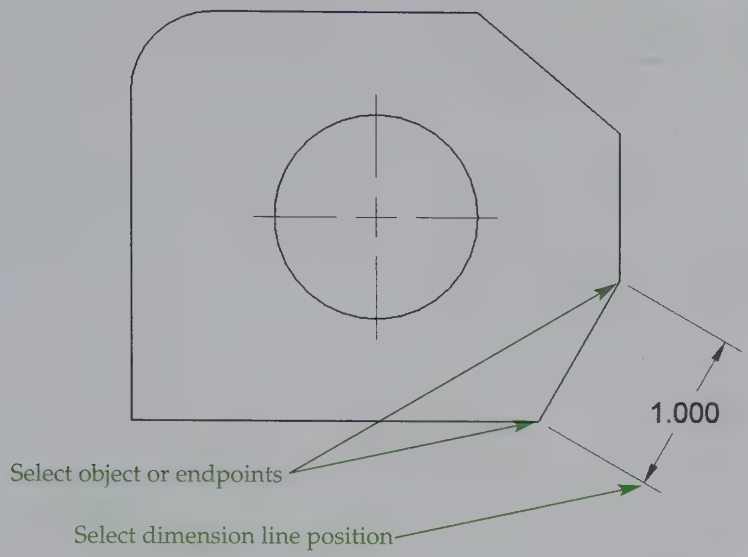
Figure 18-11.
Selecting dimension line positions and offsets specified in the **Styles and Standards Editor**.



the **Styles and Standards Editor**. When the dimension becomes dotted, pick the location to accept the specified offset. See Figure 18-11.

- **Aligned.** Typically *aligned* dimensions refer to a style of dimensioning in which the dimension text is aligned with the dimension line. However, Inventor describes an aligned dimension as a dimension in which the dimension line is parallel to an inclined angular surface. See Figure 18-12. To create an aligned dimension, first select an angled object or the endpoints of the object. Then, pull the dimension perpendicularly away from the selected object, and pick the dimension line position. An alternative method is to right-click after you initially pick the object or endpoints, and select the **Aligned** option from the **Dimension Type** cascading submenu. To generate the dimension, drag the dimension line to the desired location and pick the position. See Figure 18-12. Again, the dimension will become dotted when you drag the dimension over the location designated in the **Spacing** and **Part Offset** edit boxes of the **Dimension Styles** area, **Display** tab of the **Styles and Standards Editor**. When the dimension becomes dotted, pick the location to accept the specified offset.
- **Radial**. *Radial* dimensions are used to define the radius of an arc, such as a half circle or a fillet or round, and are identified by an R, followed by the radius value. See Figure 18-13. To create a radial dimension, select the arc you want to dimension. Then, drag the dimension text to the desired location and pick the position. See Figure 18-13. If you want to override the radial dimension style, right-click before you pick the dimension text position and select **Arrowheads Inside** or **Jogged** from the **Options** cascading submenu. You can also specify a different dimension type by selecting **Diameter**, **Angle**, **Arc Length**, or **Chord Length** from the **Dimension Type** cascading submenu, depending on the application.

Figure 18-12.
Aligned dimensioning.



- **Diameter.** *Diameter* dimensions are used to define the diameter of a circle, such as a hole, and are identified by ∅, followed by the diameter value. See Figure 18-14. Placing a diameter dimension is similar to placing an angular dimension. First, select the circle you want to dimension. Then drag the dimension text to the desired location and pick the position. See Figure 18-14. You can also override the current diameter dimension options, by right-clicking before you pick the dimension text position, and select **Leader From Center** from the **Options** cascading submenu or **Radius** from the **Dimension Type** cascading submenu, depending on the application.
- **Angular.** *Angular* dimensions are used to specify the incline of an angular surface, such as a chamfer. See Figure 18-15. Typically, angular dimension

Figure 18-13.
Radial dimensioning.

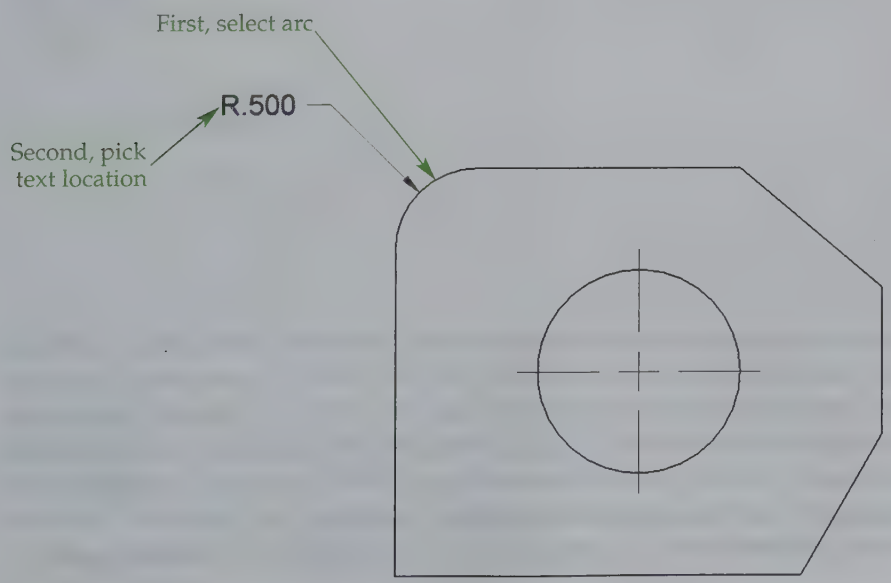


Figure 18-14.
Diameter dimensioning.

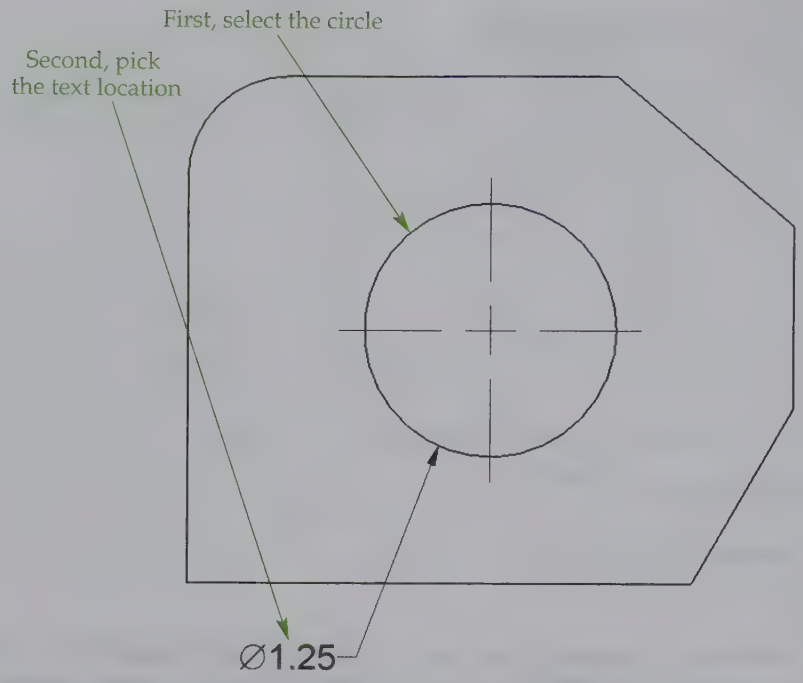
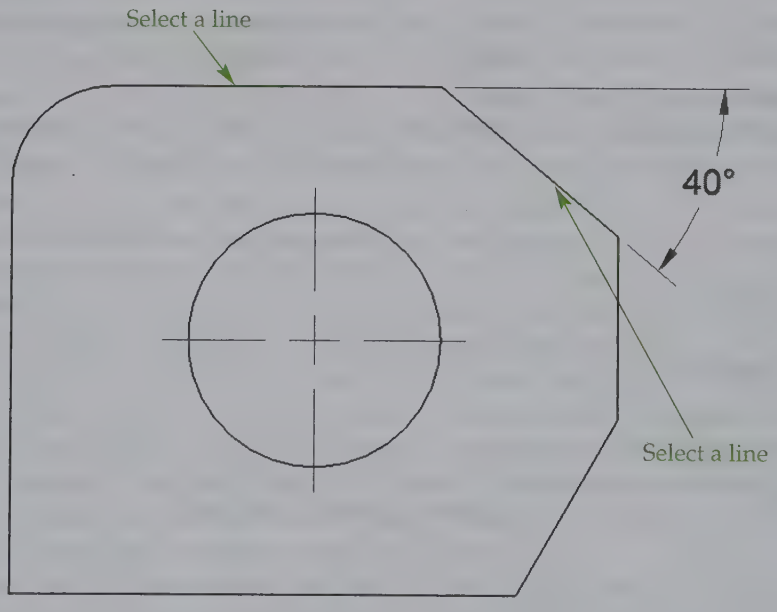
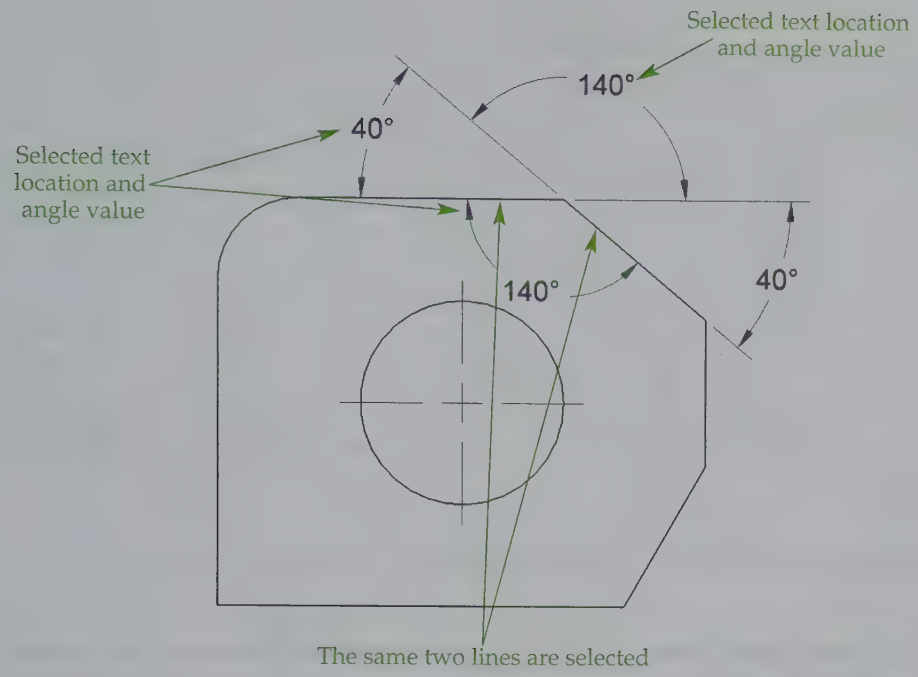


Figure 18-15.
Angular dimensioning.



values are set to degrees, for example 45°, but an angle may also be represented by degrees, minutes, and seconds, such as 25°12'36". To create an angular dimension, select one of the lines that define the angled surface, followed by the second line that defines the angled surface. To generate the dimension, drag the dimension text to the desired location and pick the position. The location you select defines the specified angle, as shown in **Figure 18-16.** Before you select the dimension text position, you can toggle between the currently

Figure 18-16.
Defining an angular dimension.



displayed dimension and the opposite angle by right-clicking and selecting the **Opposite Angle** option from the **Options** cascading submenu. As with linear and aligned dimensions, angular dimensions become dashed when you drag the dimension over the location designated in the **Spacing** and **Part Offset** edit boxes of the **Dimension Styles** area, **Display** tab of the **Styles and Standards Editor**. When the dimension becomes dashed, pick the location to accept the specified offset.

When you are finished placing general dimensions, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, or access a different tool.

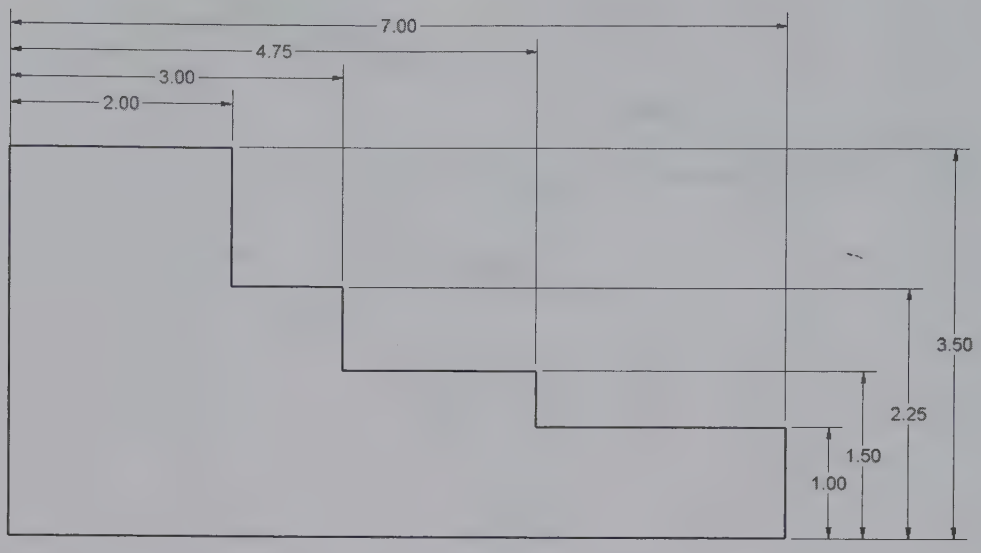


Exercise 18-4 Complete the Exercise on the Student CD.

Creating Datum Dimensions

Datum dimensioning, also referred to as *baseline dimensioning* in Inventor, is a way to dimension drawing geometry by defining the size and location of each feature in reference to an origin, also called a datum. Adjacent dimension values are staggered to reduce dimension crowding and improve clarity. See Figure 18-17. Datum dimensioning can be accomplished using the **General Dimension** tool. However, Inventor provides two tools, **Baseline Dimension Set** and **Baseline Dimension**, for adding datum dimensions to drawing view geometry. Dimensions created using the **Baseline Dimension Set** tool are linked together, while dimensions placed using the **Baseline Dimension** tool are separate from other dimensions in the group and can be individually modified if needed. Still, the process of creating datum dimensions is exactly the same whether you use the **Baseline Dimension Set** tool or the **Baseline Dimension** tool. Access the **Baseline Dimension Set** tool by picking the **Baseline Dimension Set** button

Figure 18-17.
This figure shows one set of datum dimensions for the horizontal features, and one set for the vertical features.



on the **Drawing Annotation Panel** bar; picking the **Baseline Dimension Set** button on the **Drawing Annotation Panel** toolbar; or pressing the [A] key on your keyboard. To access the **Baseline Dimension** tool, pick the **Baseline Dimension** button on the **Drawing Annotation Panel** bar, or pick the **Baseline Dimension** button on the **Drawing Annotation Panel** toolbar.

After you access the **Baseline Dimension Set** or **Baseline Dimension** tool, create datum dimensions by first selecting the common surface, or ***datum***, which is the object, often a surface, from which each dimension originates. Then, pick each of the additional objects you want to include in the dimension set. Typically the order in which you select additional features does not matter. However, the common surface must be selected first. See **Figure 18-18.** Once you select the common surface and all the geometry you want to include in the dimension group, right-click and pick the **Continue** menu option. Then, place the group of dimensions by picking the desired location of the initial dimension line. As in other dimensions, the dimension will become dotted when you drag the dimension over the location designated in the **Spacing** and **Part Offset** edit boxes of the **Dimension Styles** area, **Display** tab of the **Styles and Standards Editor**. When the dimension becomes dotted, pick the location to accept the specified offset. See **Figure 18-19.** After you pick the location for the group of dimensions, you can add dimensions to the group by picking additional features. When you are satisfied with the location and number of dimensions, you must right-click and select the **Create** menu option in order to produce the dimensions. To cancel the tool without creating dimensions, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, or access a different tool.

Datum dimensions created using the **Baseline Dimension** tool are totally independent from other dimensions, and can be modified like any other linear dimension. However, dimensions placed using the **Baseline Dimension Set** tool are controlled by the entire group of dimensions. You can include additional geometry in a dimension set by right-clicking on the existing dimensions and selecting the **Add Member** pop-up menu option. Then, select the geometry you want to add to the dimensions. You can also remove dimensions from a set of datum dimensions by right-clicking on the

Figure 18-18.
Creating datum, or baseline, dimensions.

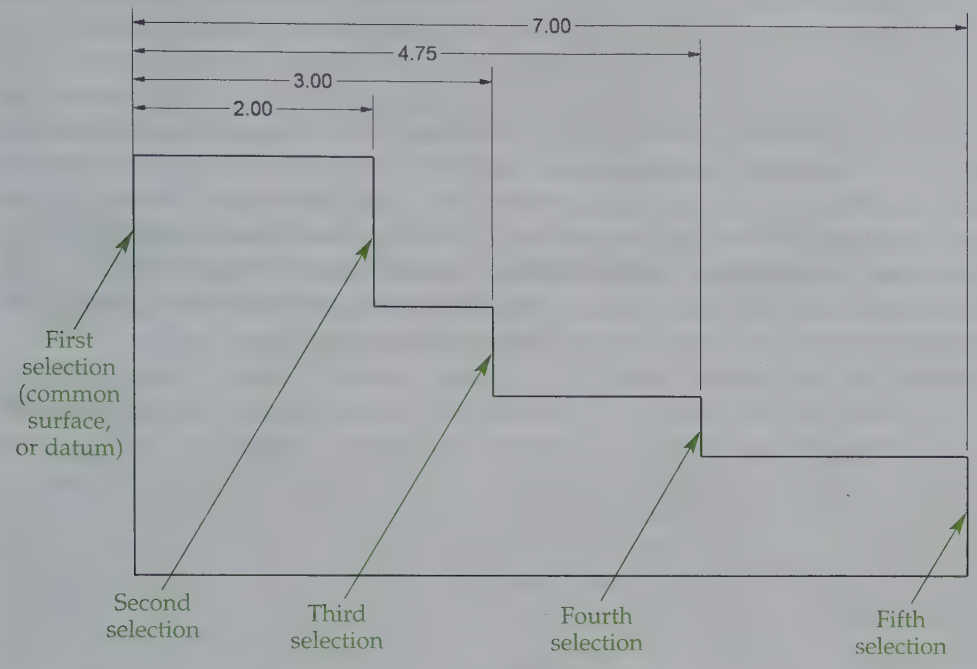
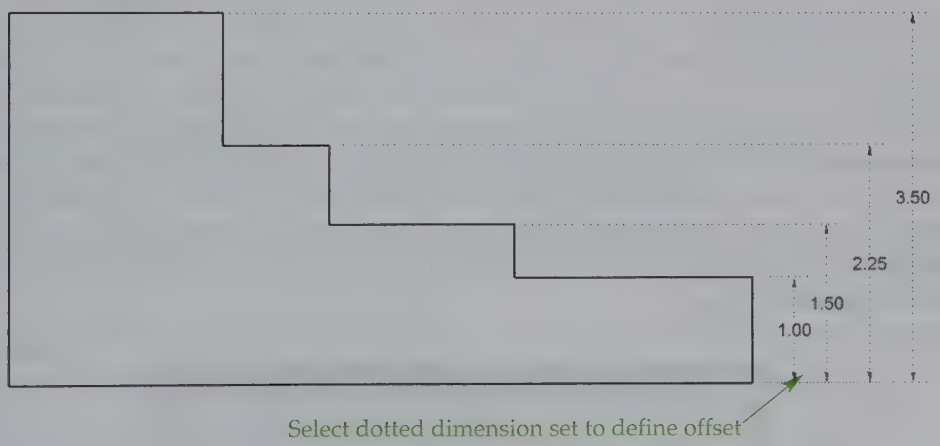
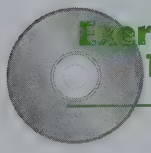


Figure 18-19.
Selecting datum dimension line positions and offsets specified in the **Styles and Standards Editor**.



dimension you want to remove and selecting the **Delete Member** pop-up menu option. Finally, if you want to detach geometry from a set of datum dimensions without actually deleting the dimension, right-click on the dimension you want to detach, and select the **Detach Member** pop-up menu option.



Exercise 18-5 Complete the Exercise on the Student CD.

Adding Text

Often you may need to apply specific or general notes or some type of additional text to a drawing, without opening a new sketch or adding a draft view. ***Specific Notes***, also called ***local notes***, are notes that apply to a specific feature or features on the drawing. ***General Notes*** apply to the entire drawing and are usually placed together in a location on the drawing such as the lower-left corner, upper-right corner, or near the title block. Inventor includes a drawing annotation **Text** tool to place note text. The **Text** tool available in the drawing annotation environment functions exactly the same as the **Text** tool available in the sketch environments described throughout this book. Text properties are controlled by text styles specified in the **Styles and Standards Editor**. You can use the default text styles or override many format options when you place text. To create text, access the **Text** tool by picking the **Text** button on the **Drawing Annotation Panel** bar, or picking the **Text** button on the **Drawing Annotation Panel** toolbar.

Once you access the **Text** tool, use the positioning cursor to select a point, or hold-down the left mouse button and drag the cursor to create a window on the sheet. See Figure 18-20. The point you select, or window you create, defines the location and extents of the text. After you designate this location, the **Format Text** dialog box is displayed. See Figure 18-21. The **Format Text** dialog box functions very much the same in the drawing annotation environment as it does in the sketch environment. For more information regarding text and the **Format Text** dialog box, refer to previous chapters. When you finish defining options in the **Format Text** dialog box, pick the **OK** button to generate the text. The **Text** tool will still be active, allowing you to continue placing

Figure 18-20.
Selecting a text location.

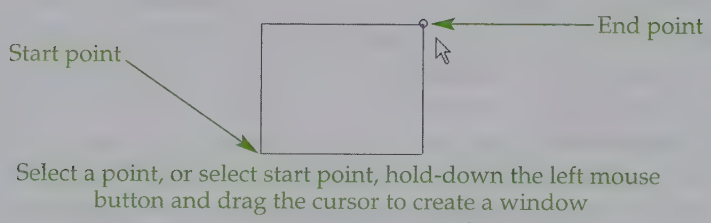
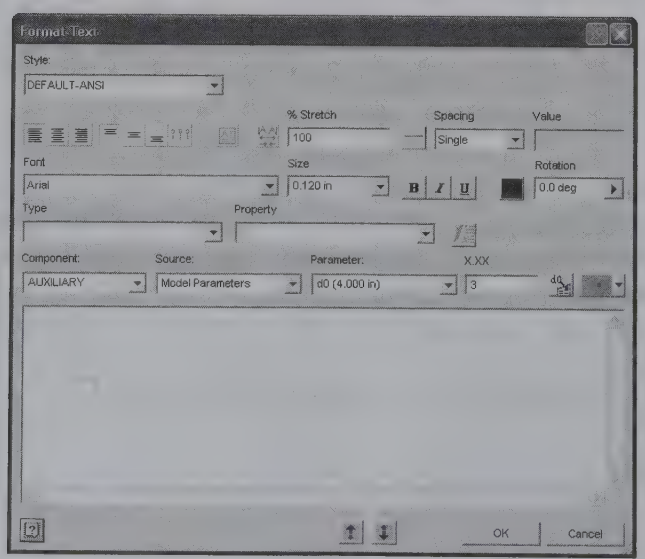


Figure 18-21.
The **Format Text** dialog box.



text on the drawing. To exit the **Text** tool, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, or access a different tool.

NOTE

Once text, such as a general note, is placed, you can rotate the text 90° by right-clicking and selecting either the **Rotate 90 CW** or **Rotate 90 CCW** pop-up menu option.

Exercise 18-6

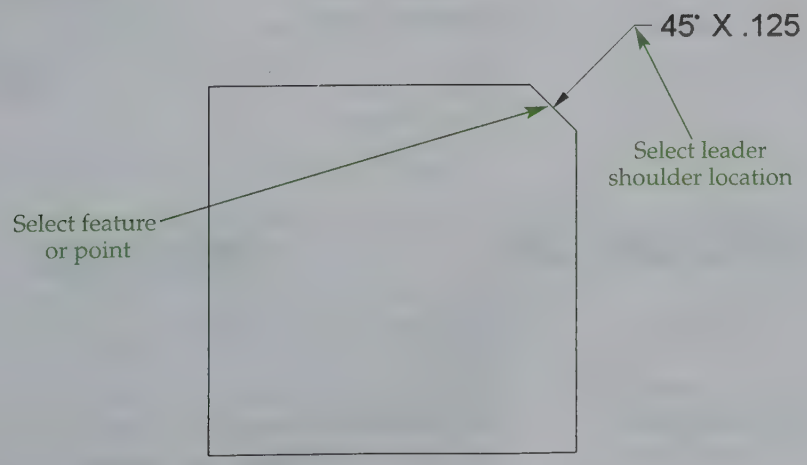
Complete the Exercise on the Student CD.

Placing Leader Notes

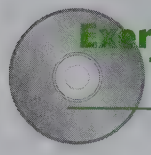
A specific, or local, note applies to a specific feature or set of features on the drawing. A specific note is often connected to a feature using a leader line, or leader, as they are typically called. A *leader* connects to the beginning or the end of a note with a horizontal shoulder and is drawn at an angle where it is capped on the end with an arrowhead that touches a feature. The **General Dimension** tool allows you to dimension circular features with leaders, the **Hole/Thread Notes** tool can be used to dimension holes and threads, and the **Chamfer Notes** tool can be used to dimension chamfers. However, you can dimension all other types of specific geometry with leaders, using the **Leader Text** tool, which is similar to the **Text** tool described earlier, but includes a leader. See Figure 18-22. To access the **Leader Text** tool, pick the **Leader Text** button on the **Drawing Annotation Panel** bar, or pick the **Leader Text** button on the **Drawing Annotation Panel** toolbar.

Once you access the **Leader Text** tool, select a feature, or a feature point, to which you want to add a leader and text, or pick a point in the drawing sheet. Then pick the desired location for leader shoulder. See Figure 18-22. After you select appropriate drawing geometry or a point on the drawing sheet and locate the text leader shoulder, press the [Enter] key on your keyboard or right-click and select the **Continue** menu option to display the **Format Text** dialog box. The **Format Text** dialog box displayed when placing leader text is exactly the same as the **Format Text** dialog box displayed when placing text without a leader. See Figure 18-21. When you finish defining options

Figure 18-22.
Placing a text leader.



in the **Format Text** dialog box, pick the **OK** button to generate the leader and the text. The **Leader Text** tool will still be active, allowing you to continue placing leaders on the drawing. To exit the **Leader Text** tool, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, or access a different tool.



Complete the Exercise on the Student CD.

Adding Hole and Thread Notes

Hole and thread notes are normally dimensioned to the feature using a leader line. Holes are dimensioned using diameter dimensions with the diameter symbol preceding the dimension value, as in ∅1.625. Thread notes are commonly connected to the thread representation using a leader line and always follow the same format. Figure 18-23 shows inch and metric thread note convention.

Inventor provides you with the **Hole/Thread Notes** tool, which allows you to fully dimension holes, holes with threads, and threaded shafts. The **Hole/Thread Notes** tool automatically dimensions information such as countersink, counter bore, depth, tap, thru, and any other hole and thread information that corresponds to the model. See Figure 18-24. To access the **Hole/Thread Notes** tool, pick the **Hole/Thread Notes** button on the **Drawing Annotation Panel** bar, or pick the **Hole/Thread Notes** button on the **Drawing Annotation Panel** toolbar.

After you access the **Hole/Thread Notes** tool, dimension a hole or threaded stud by first selecting the feature. Then, pick the position of the dimension text. Continue adding hole and thread notes as needed. See Figure 18-24. When you are finished, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, or access a different tool.

Figure 18-23.
The thread notes for Unified and American National Threads are written in one format, while metric threads are written in another format.

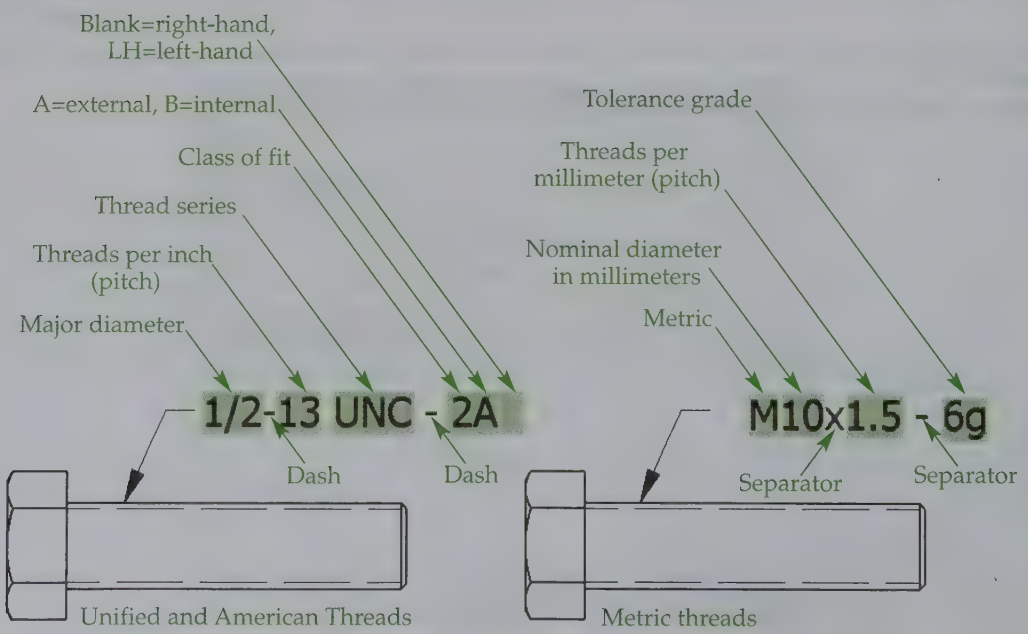
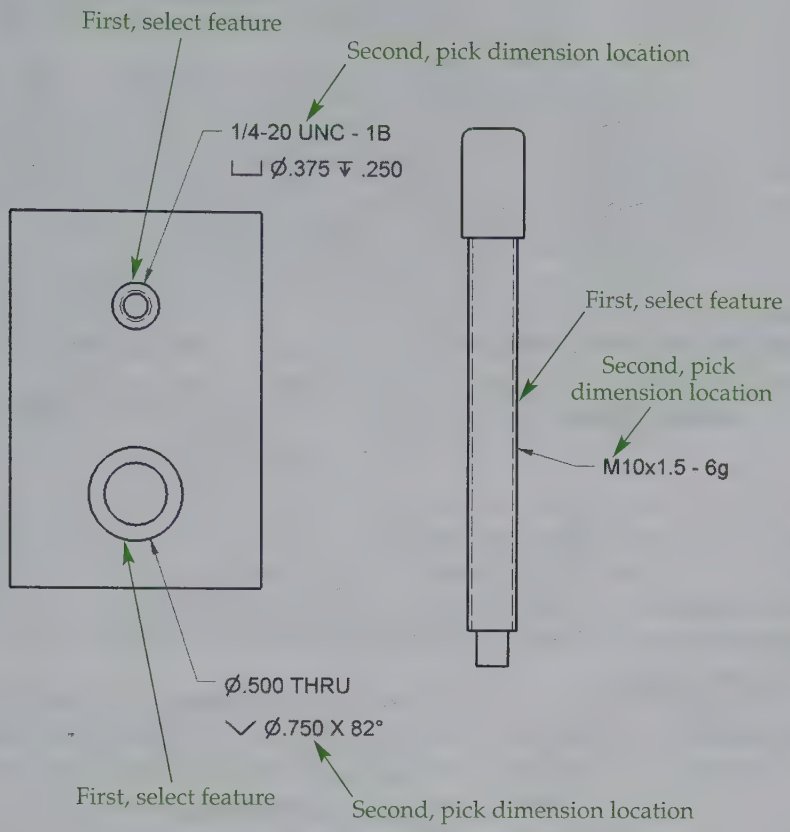


Figure 18-24.
Using the **Hole/Thread Notes** tool.



NOTE

To redefine hole note properties for a specific hole/thread note, right-click on an existing hole/thread note and pick the **Edit Hole Note...** pop-up menu option. Selecting **Edit Hole Note...** opens the **Edit Hole Note** dialog box, which contains many of the same options found in the **Notes and Leaders** tab of the **Styles and Standards Editor, Dimension Style** area.

Exercise 18-8 Complete the Exercise on the Student CD.

Using the Chamfer Note Tool

Chamfers can be dimensioned a variety of ways, depending on the chamfer size and the angle of the inclined surface. 45° chamfers are usually dimensioned with a leader giving the angle and linear dimension, or with two linear dimensions. See Figure 18-25. The **Chamfer Note** tool can be used to dimension 45° chamfers, using a leader and either an angle and a linear distance, or two linear distances, depending on the dimension style settings. To access the **Chamfer Note** tool, pick the **Chamfer Note** button on the **Drawing Annotation Panel** bar, or pick the **Chamfer Note** button on the **Drawing Annotation Panel** toolbar.

After you access the **Chamfer Note** tool, dimension a chamfer by selecting the inclined surface followed by the adjacent reference edge. Then, pick the position of the dimension text. Continue adding chamfer notes as needed. See **Figure 18-26**. When you are finished, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, or access a different tool.

Chamfers other than 45° must have either the angle and a linear dimension or two linear dimensions placed on the view. See **Figure 18-27**. Using the **Chamfer Note** tool to dimension a non-45° chamfer leaves the chamfer undefined, by not specifying whether the angle is horizontal or vertical. As a result, it is best to use linear dimensions or a linear and angular dimension to define non-45° chamfers.

Figure 18-25.
Common practices for dimensioning chamfers.

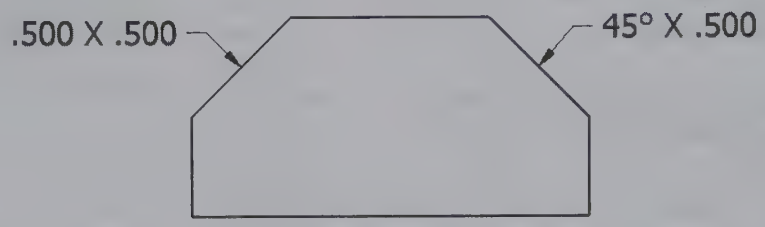


Figure 18-26.
Using the **Chamfer Note** tool.

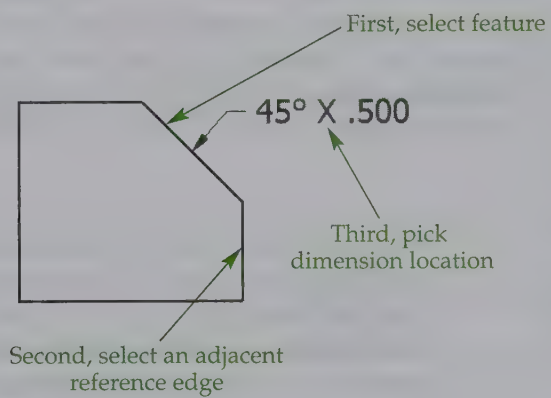
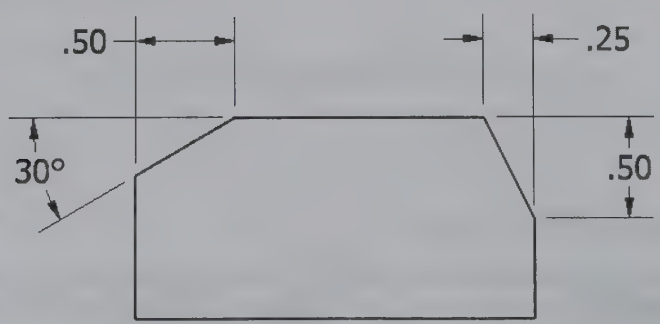


Figure 18-27.
An angle and a linear dimension identify the chamfer on the left. Two linear dimensions identify the chamfer on the right.





Exercise 18-9 Complete the Exercise on the Student CD.

Placing Surface Texture Symbols

Surface finish refers to the allowable roughness, waviness, law, and flaws on a surface. These imperfections are obtained by machining, grinding, honing, or lapping processes. The surface finish symbol is a V shape that contains values related to the finish characteristics. The Inventor **Surface Texture Symbol** tool allows you to place the specified surface finish characteristic values and surface finish symbol at the desired location, with or without a leader line. See Figure 18-28. To access the **Surface Texture Symbol** tool, pick the **Surface Texture Symbol** button on the **Drawing Annotation Panel** bar, or pick the **Surface Texture Symbol** button on the **Drawing Annotation Panel** toolbar.

Once you access the **Surface Texture Symbol** tool, use the cursor to select the feature, or point, to which you want to add surface texture information. Then, pick a location for the surface texture symbol and right-click to select the **Continue** menu option and display the **Surface Texture** dialog box, or press the [Enter] key to place the surface texture symbol on the selected object and display the **Surface Texture** dialog box. See Figure 18-29. Use the following options in the **Surface Texture** dialog box to fully define the finish of a part surface:

NOTE



The options in the **Surface Texture** dialog box will vary depending on the settings you specify in the **Surface Texture Style** area of the **Styles and Standards Editor**.

- **Surface Type.** This area contains buttons that allow you to define the type of surface finish symbol. You can select the **Standard Surface Finish Symbol** button, **Removal of Material Required** button, or the **Removal of Material Prohibited** button, depending on the application.
- **Miscellaneous.** This area contains the **Force Tail** button, which allows you to force a tail on the surface texture symbol. Other buttons specified in the **Surface Texture Style** area of the **Styles and Standards Editor** are also located in the **Miscellaneous** area including **Majority** and **All-Around** buttons.
- **A′.** This edit box allows you to define the maximum roughness height, or if you do not specify a minimum roughness height, use this edit box to define the roughness average, or Ra.
- **A.** This edit box allows you to define the minimum roughness height, when a maximum roughness height is set in the **A′** edit box.
- **B.** Use this edit box to enter the waviness of the surface texture.

Figure 18-28.
Examples of surface finish definitions.

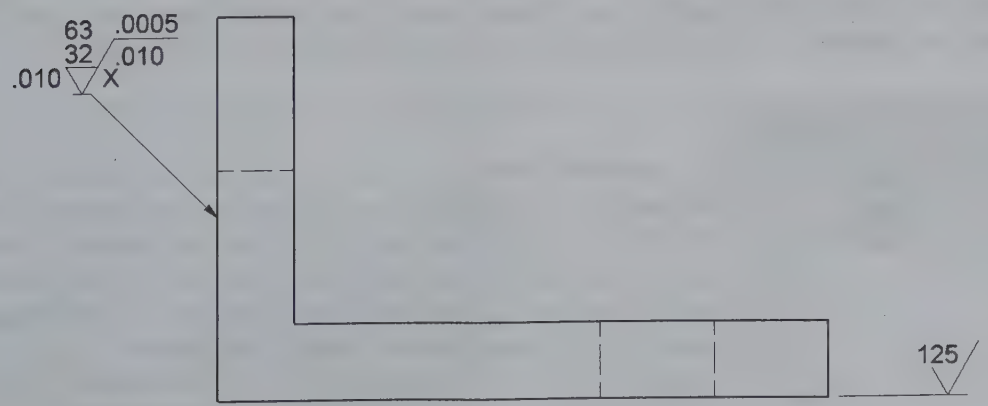
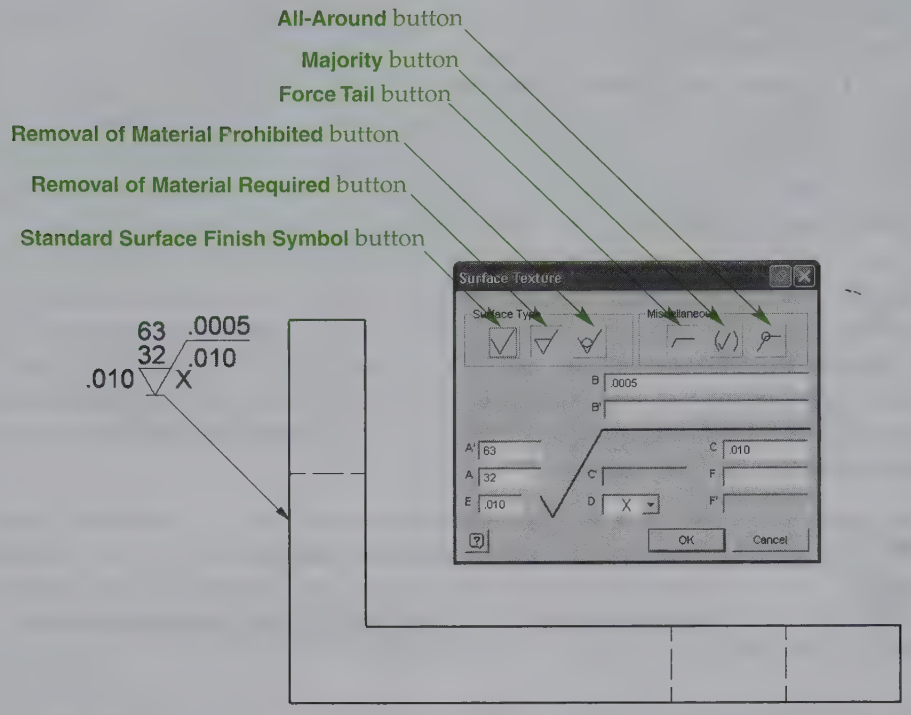


Figure 18-29.
Placing a surface texture symbol using the **Surface Texture** dialog box.



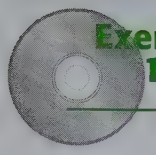
- **B′.** This edit box is available when a value is added to the **B** edit box, and is used to identify an additional waviness value.
- **C′.** Use this edit box to enter the roughness sampling length, or cutoff value.
- **C.** This edit box is used to also define the roughness sampling length, or cutoff value, or an additional value.
- **D.** Use this drop-down list to select the desired machining lay. You can choose from parallel (=) or perpendicular (b) to edge, angular in both directions (X), multidirectional (M), circular relative to center (C), radial relative to center (R), or nondirectional particulate (P).
- **F.** This edit box is used to specify a roughness other than Ra, which is defined in the **A′** edit box.
- **F′.** This edit box is used to specify a minimum roughness other than Ra, when a value is entered in the **F** edit box.
- **E.** Use this edit box to identify the machining allowance, which specifies the amount of stock to be removed by machining.

When you have fully defined the surface texture specifications in the **Surface Texture** dialog box, pick the **OK** button to create the surface texture symbol. Continue adding surface finish symbols as needed, and when you are finished, right-click and select the **Done** menu option, press the [Esc] key, or access a different tool, to exit the tool.

PROFESSIONAL TIP



To generate surface texture symbols that include an extension line, highlight the surface texture symbol you want to move or pick the surface texture symbol. Then move your cursor over the green dot that intersects the leader line and leader shoulder. When you see the **Move** icon, pick the green dot and drag the symbol to the desired location. The leader should change into an extension line.



Complete the Exercise on the Student CD.

Ordinate Dimensioning

Ordinate dimensioning is also known as *arrowless dimensioning*, because traditional dimension lines and arrowheads are not used. Ordinate dimensioning places dimension values at the end of extension lines, providing coordinates from established datums that are often the corner of the part or the axis of a feature. See Figure 18-30. Using ordinate dimensioning techniques, features such as holes and radii can be dimensioned in a traditional manner, or letters can be placed on the drawing identifying each feature and coordinating the letter to a table where the dimensional value is given. For example, all holes of one diameter are labeled A, and holes of another diameter are labeled B, and so on. Inventor provides two tools, **Ordinate Dimension Set** and **Ordinate Dimension**, for adding ordinate dimensions to drawing view geometry. Dimensions created using the **Ordinate Dimension Set** tool are linked together, while dimensions placed using the **Ordinate Dimension** tool are separate from other dimensions in the group and can be individually modified if needed.

Access the **Ordinate Dimension Set** tool by picking the **Ordinate Dimension Set** button on the **Drawing Annotation Panel** bar; picking the **Ordinate Dimension Set** button on the **Drawing Annotation Panel** toolbar; or pressing the [O] key on your keyboard.

After you access the **Ordinate Dimension Set** tool, create ordinate dimensions by first selecting the origin. The origin establishes the 0,0 point, or datum, which is typically the corner of the part, or the axis of a feature. Then, pick each of the additional objects you want to include in the dimension set. Typically, the order in which you select additional features does not matter. However, the common surface must be selected first. See Figure 18-31. Once you select the origin and all the geometry you want to include in the dimension group, right-click and pick the **Continue** menu option. Then, place the group of dimensions by picking the desired location of the ini-

Figure 18-30.
An example of ordinate dimensioning.

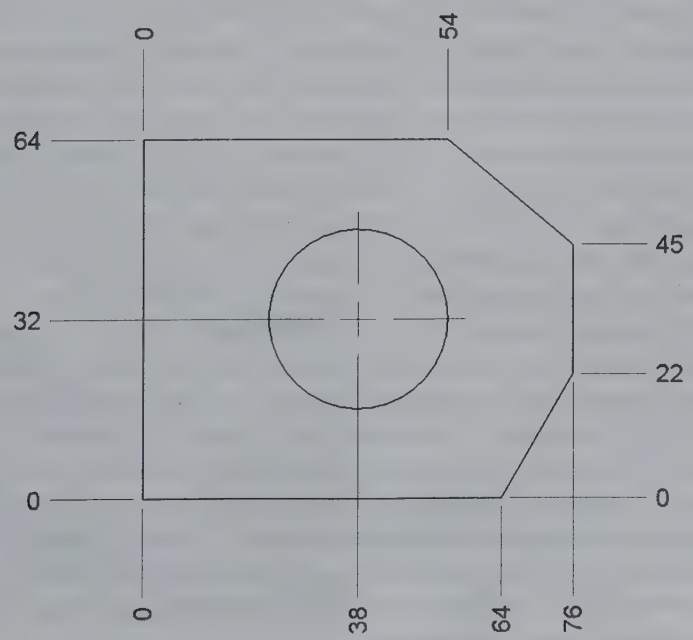
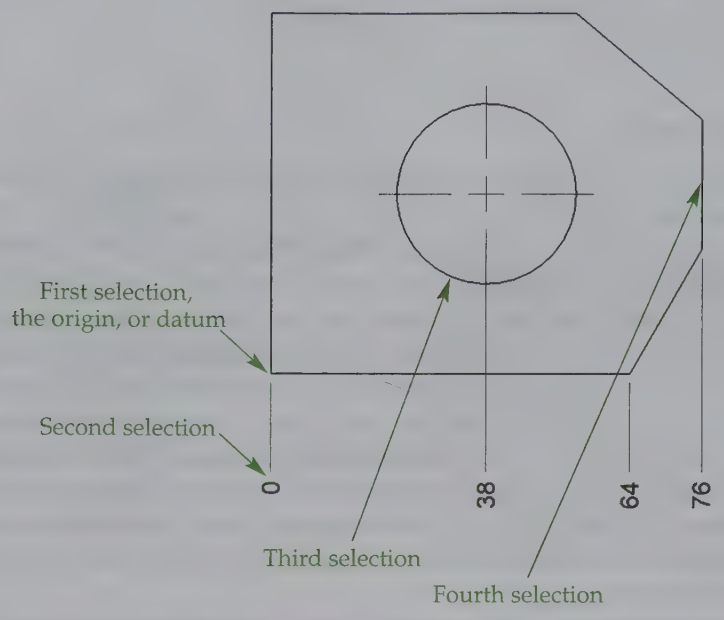


Figure 18-31.
Creating ordinate dimensions using the **Ordinate Dimension Set** tool.

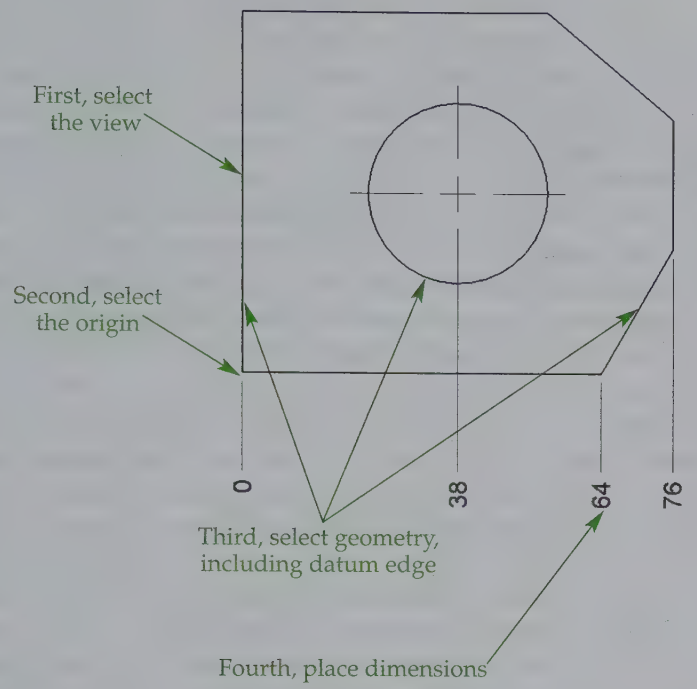


tial dimension line. After you pick the location for the group of dimensions, you can add dimensions to the group by picking additional features. While placing ordinate dimensions, you can right-click and select any of the choices in the **Options** cascading submenu to apply the options, depending on your application. When you are satisfied with the location and number of dimensions, you must right-click and select the **Create** menu option in order to produce the dimensions. To cancel the tool without creating dimensions, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, or access a different tool.

Access the **Ordinate Dimension** tool by picking the **Ordinate Dimension** button on the **Drawing Annotation Panel** bar, or picking the **Ordinate Dimension** button on the **Drawing Annotation Panel** toolbar. The first step in creating a set of ordinate dimensions is to select the view you want to dimension. Then, select the origin to establish the 0,0 point, or datum, which is typically the corner of the part, or the axis of a feature. Next, pick the initial dimension value location, which should be 0, drag the value to the desired location, and select the position. Continue selecting additional object endpoints to dimension. See Figure 18-32. Once you select all the geometry you want to dimension along a specified plane, right-click and select the **Create** menu option to generate the ordinate dimensions. You will notice after you create the dimensions, the **Ordinate Dimension** tool is still active. You can repeat the process of selecting the view, followed by the datum, and then the other view geometry to place additional view dimensions. You may also choose to exit the tool by right-clicking and selecting the **Done** menu option or pressing the [Esc] key on your keyboard.

Ordinate dimensions created using the **Ordinate Dimension** tool are totally independent from other dimensions, and can be modified separately using basic editing options, if needed. However, dimensions placed using the **Ordinate Dimension Set** tool are controlled by the entire group of dimensions. You can include additional geometry in a dimension set by right-clicking on the existing dimensions and selecting the **Add Member** pop-up menu option. Then, select the geometry you want to add to the dimensions. You can also remove dimensions from a set of datum dimensions by right-clicking on the dimension you want to remove, and selecting the **Delete Member** pop-up menu option. Several other ordinate dimension variables can also be controlled by right-clicking on an ordinate dimension set.

Figure 18-32.
Creating ordinate dimensions using the **Ordinate Dimension** tool.



NOTE

You can turn off the origin indicator visibility by right-clicking on the origin indicator or an ordinate dimension value, and selecting the **Hide Origin Indicator** pop-up menu option. The same origin and origin indicator is used for ordinate and tabular dimensioning.

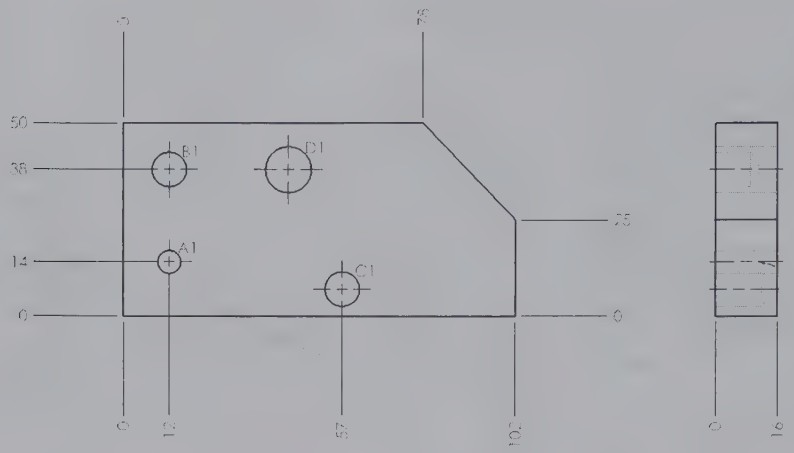
Exercise 18-11

Complete the Exercise on the Student CD.

Tabular Dimensioning

Tabular dimensioning is a type of arrowless, ordinate dimensioning that involves a system in which coordinate dimensions and size dimensions are given in a table, correlating features on the drawing with a hole tag. See Figure 18-33. Inventor provides three tools, **Hole Table - Selection**, **Hole Table - View**, and **Hole Table - Selected Feature**, for placing tabular dimensions. The **Hole Table - Selection** tool allows you to create a dimension table by selecting specific drawing view features, the **Hole Table - View** allows you to create a table by picking a drawing view, and the **Hole Table - Selected Feature** tool allows you to select a single hole and create a dimension table that consists of all the holes of the same selected hole type. Like other dimensioning tools, hole table tools do not fully dimension a feature. For example, you must still use the **Center Mark** tool to place hole center marks, and the **Ordinate Dimension Set** or **Ordinate Dimension** tools to place ordinate dimensions, as shown in Figure 18-33. Figure 18-34 displays the results of using hole table tools without adding any other dimension information.

Figure 18-33.
An example of tabular dimensioning.



HOLE TABLE			
HOLE	XDIM	YDIM	DESCRIPTION
A1	12	14	Ø6 THRU
B1	12	38	Ø9 ↧ 9
C1	57	7	Ø9 ↧ 12
D1	43	38	Ø12 THRU

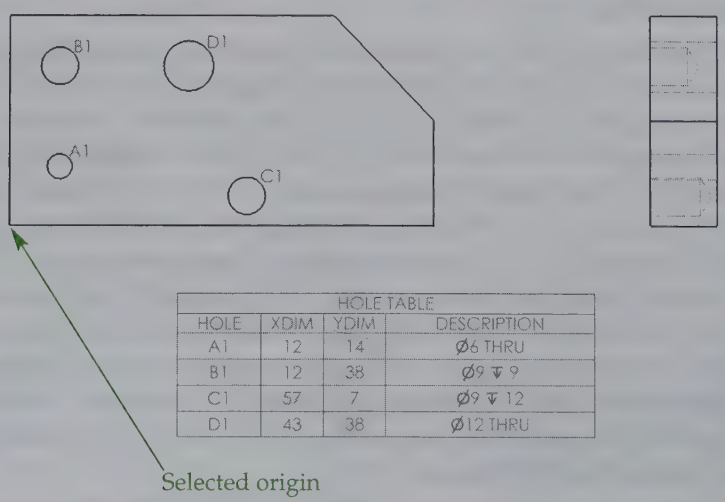
NOTE



In the **Drawing Annotation Panel** toolbar and panel bar, the **Hole Table - Selection**, **Hole Table - View** and **Hole Table - Selected Feature** tools are located in a flyout button.

To access the **Hole Table - Selection** tool, pick the **Hole Table - Selection** button on the **Drawing Annotation Panel** bar; pick the **Hole Table - Selection** button on the **Drawing Annotation Panel** toolbar; or pick the **Hole Table - Selection** button on the **Drawing Hole Table** toolbar.

Figure 18-34.
The effects of using the various hole table tools, without adding any other dimension information.



HOLE TABLE			
HOLE	XDIM	YDIM	DESCRIPTION
A1	12	14	Ø6 THRU
B1	12	38	Ø9 ↧ 9
C1	57	7	Ø9 ↧ 12
D1	43	38	Ø12 THRU

Once you access the **Hole Table - Selection** tool, the first step in creating a hole table is to select the drawing view, followed by selecting the origin, unless an origin has already been added. Just as in ordinate dimensioning, the origin establishes the 0,0 point, or datum, which is typically the corner of the part, or the axis or center of a feature. Next, select each of the holes you want to include in the table. The order in which you select holes determines the hole tag designation and hole table order. For example, the first hole you select is labeled A1, and the second hole you select is labeled B1, assuming the hole types are different. Once you select each of the holes that you want to display in the table, right-click and pick the **Create** menu option. Then move the table to the desired location and pick the position. See Figure 18-35.

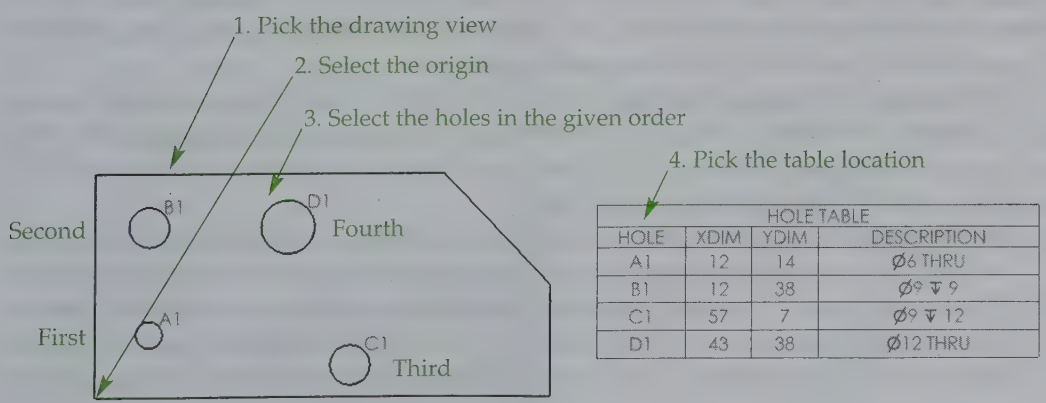
To activate the **Hole Table - View** tool, pick the **Hole Table - View** button on the **Drawing Annotation Panel** bar; pick the **Hole Table - View** button on the **Drawing Annotation Panel** toolbar; or pick the **Hole Table - View** button on the **Drawing Hole Table** toolbar.

Once you access the **Hole Table - View** tool, dimensioning a drawing view is a three-step process. The first step is to select the drawing view, which automatically selects all the available holes in the view. The second step is to pick the origin, unless an origin has already been added. Again, the origin establishes the 0,0 point, or datum, which is typically the corner of the part, or the axis or center of a feature. The third and final step is to move the table to the desired location, and pick the table position. See Figure 18-36.

To activate the **Hole Table - Selected Feature** tool, pick the **Hole Table - Selected Feature** button on the **Drawing Annotation Panel** bar; pick the **Hole Table - Selected Feature** button on the **Drawing Annotation Panel** toolbar; or pick the **Hole Table - Selected Feature** button on the **Drawing Hole Table** toolbar.

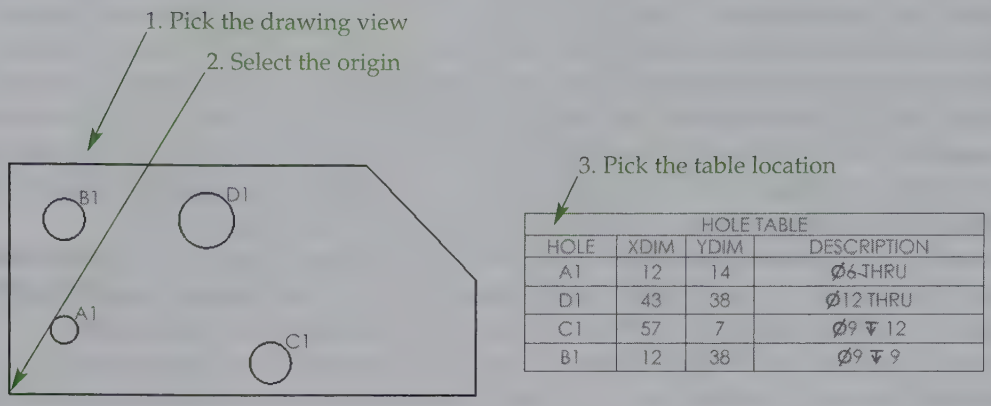
Once you access the **Hole Table - Selected Feature** tool, the first step in creating a hole table is to select the drawing view, followed by selecting the origin, unless an origin has already been added. Again, the origin establishes the 0,0 point, or datum, which is typically the corner of the part, or the axis or center of a feature. Next, select the hole features that you want to include in the table. As previously mentioned, when using the **Hole Table - Selected Feature** tool, you need to select only one instance of the hole on the drawing, because each additional hole of the same type is automatically selected. If you pick more than one hole type, the order in which you choose holes determines the hole tag designation and hole table order. For example, the first hole type you select is labeled A, and the second hole you select is labeled B. Once you select each of the hole types that you want to display in the table, right-click and pick the **Create** menu option. Then, move the table to the desired location, and pick the position. See Figure 18-37.

Figure 18-35.
Tabular dimensioning using the **Hole Table - Selection** tool.



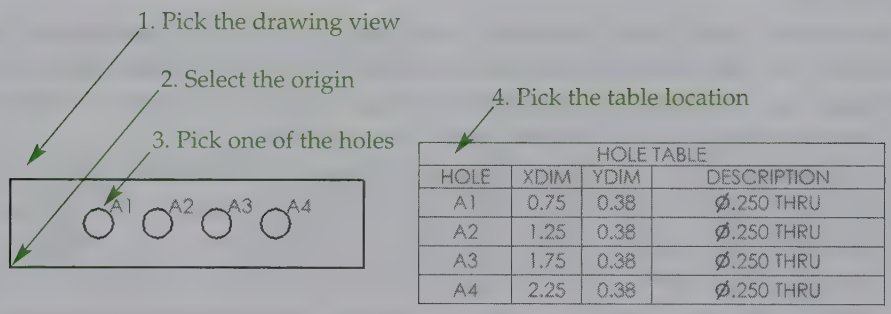
HOLE TABLE			
HOLE	XDIM	YDIM	DESCRIPTION
A1	12	14	Ø6 THRU
B1	12	38	Ø9 ↧ 9
C1	57	7	Ø9 ↧ 12
D1	43	38	Ø12 THRU

Figure 18-36.
Tabular dimensioning using the **Hole Table - View** tool.



HOLE TABLE			
HOLE	XDIM	YDIM	DESCRIPTION
A1	12	14	Ø6 THRU
D1	43	38	Ø12 THRU
C1	57	7	Ø9 ↧ 12
B1	12	38	Ø9 ↧ 9

Figure 18-37.
Tabular dimensioning using the **Hole Table - Selected Feature** tool.



HOLE TABLE			
HOLE	XDIM	YDIM	DESCRIPTION
A1	0.75	0.38	Ø.250 THRU
A2	1.25	0.38	Ø.250 THRU
A3	1.75	0.38	Ø.250 THRU
A4	2.25	0.38	Ø.250 THRU

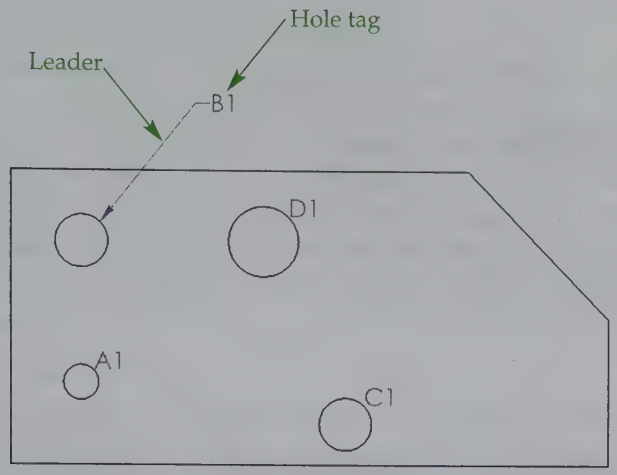
Once you generate tabular dimensions using the **Hole Table - Selection**, **Hole Table - View**, and **Hole Table - Selected Feature** tools, there are a number of options for modifying and adding dimension information, depending on the application and desired dimension appearance. Default hole table settings should be predefined in the **Styles and Standards Editor**. However, the following tabular dimension items can be redefined using the techniques described.

You can turn off the origin indicator visibility by right-clicking on the hole table and selecting **Origin** from the **Visibility** cascading submenu, or by right-clicking on the origin indicator itself, and selecting the **Hide Origin Indicator** menu option. The same origin and origin indicator is used for ordinate and tabular dimensioning.

For some applications, the default hole tags created when you place tabular dimensions may require modification. The following information describes possible hole tag redefinition options you may want to explore:

- **Move tags.** To move a hole tag, place your cursor over the tag or select the tag to display the green dot and the **Move** icon. Then drag the green dot and the tag to the desired location. If you pull the tag too far from the hole, a leader is automatically created. See Figure 18-38.
- **Add tag leaders.** As mentioned, a leader can be added to a hole tag by dragging the existing tag away from the hole. You can also place the leader by right-clicking on the hole tag and selecting the **Show Leader** pop-up menu option, but you will still need to drag the tag to a new location for leader clarity. Conversely, to hide a tag leader, right-click on the leader or tag and select the

Figure 18-38.
An example of a hole tag with an attached leader.



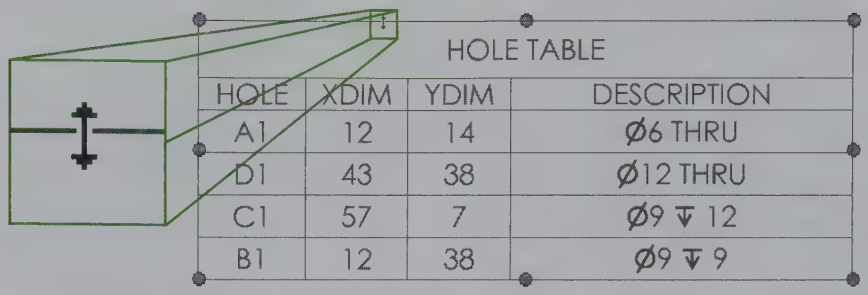
Hide Leader pop-up menu option, but again you will have to drag the tag back to its original location.

- **Hide tags.** You can hide a single hole tag by right-clicking on the tag in the drawing and selecting the **Hide Tag** pop-up menu option, or by right-clicking on a tag in the hole table and deselecting **Tag** from the **Visibility** cascading submenu. Once hidden, to redisplay a tag you must right-click on a tag in the hole table and select **Tag** or **Show All Tags** from the **Visibility** cascading submenu. You also have the option of hiding all hole tags by right-clicking on the hole table and selecting **Hide All Tags** from the **Visibility** cascading submenu, and redisplaying all tags by right-clicking on the hole table and selecting **Show All Tags** from the **Visibility** cascading submenu.
- **Edit tags.** To modify the definition of a tag, right-click on the tag in the drawing and select the **Edit Tag...** pop-up menu option, or right-click on a tag in the hole table and select **Edit Tag** from the **Edit** cascading submenu. Both of these operations open the **Format Text** dialog box for the selected tag, allowing you to edit or add tag information.

For some applications the default hole table created when you place tabular dimensions may require modification or added content. In addition to the options previously described, the following information describes possible hole table redefinition options you may want to explore:

- **Resize rows and columns.** A hole table *row* is the horizontal area that contains information about each hole, which means every hole has its own row. A hole table *column* is the vertical area that contains a specific set of tabular dimension information, such as size. You can expand and contract rows and columns by moving your cursor over a row or column line until you see the **Resize** icon, and then dragging the row or column to the desired location. See Figure 18-39.
- **Resorting table contents.** Often, hole tags are modified according to specific tabular dimensioning requirements. For example, you may change hole tags A1, B1, C1, and D1 to A1, A2, B1, and C1. In order to represent the changes correctly in the hole table and display the holes in chronological order, pick the **Resort Table** option from the **Table** cascading submenu.
- **Split a table.** In some situations, you may need to divide a hole table into two or more separate tables. To accomplish this task, right-click on the row at which you want the division to occur, and pick the **Split Table** option from the **Table**

Figure 18-39.
Manipulating hole table row and column sizes.



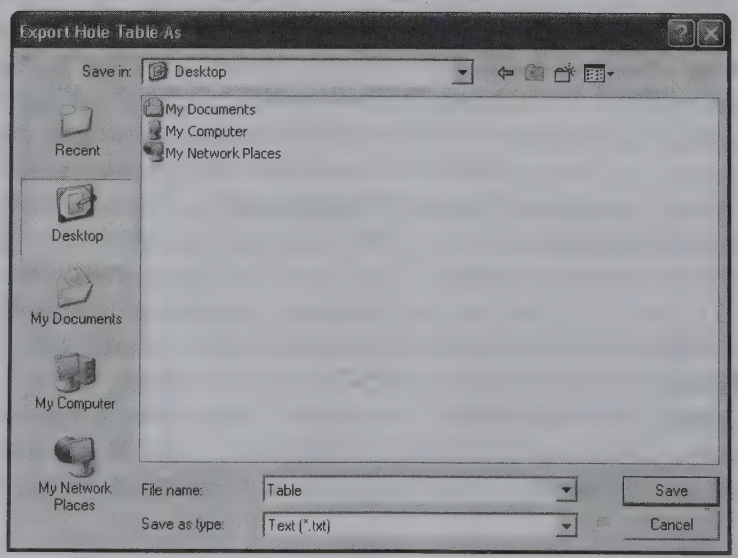
HOLE TABLE

HOLE	XDIM	YDIM	DESCRIPTION
A1	12	14	Ø6 THRU
D1	43	38	Ø12 THRU
C1	57	7	Ø9 ↧ 12
B1	12	38	Ø9 ↧ 9

cascading submenu. When a table is split, a new table is created that can be moved and manipulated independent of the other hole table or tables.

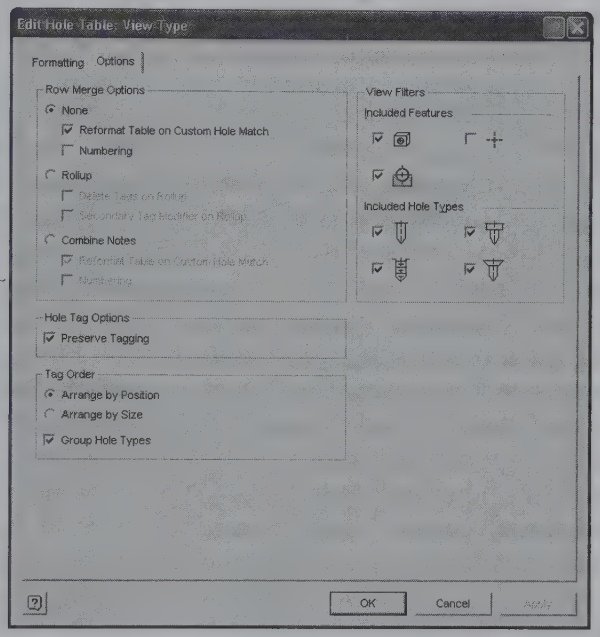
- **Export a table.** To export, or save, the hole table to an external, or non-Inventor file, right-click on the hole table in the graphics window or browser, and select the **Export Table** option from the **Table** cascading submenu. When you select the **Export Table** option, the **Export Hole Table As** dialog box is displayed. See Figure 18-40. The **Export Hole Table As** dialog box functions much like any other **Save As** dialog box, allowing you to select a location for the hole table file, name the file, and select the type of file you want to save the table as. A hole table can be exported as a .txt or .csv file.
- **Add and remove hole table rows.** To add an additional hole to the hole table without deleting and recreating tabular dimensions, right-click on the hole table in the graphics window or browser, and select the **Add Hole** option from the **Row** cascading submenu. Once you select the **Add Hole** option, you can pick additional holes from the drawing view associated with the hole table. Finally, right-click and select the **Create** pop-up menu option to generate a hole tag, and add the hole to the hole table as a new row. Conversely, you can remove a hole from the table by right-clicking on the hole name you want to remove from the hole table, and selecting the **Delete Hole** option from the **Row** cascading submenu.

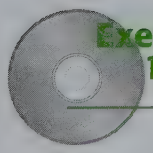
Figure 18-40.
The **Export Hole Table As** dialog box.



- **Edit hole table information.** The contents of the hole table, including individual tags, hole notes and descriptions can be modified by right-clicking on the item in the hole table that requires editing and selecting the appropriate option from the **Edit** cascading submenu. Many of these options have already been described.
- **Edit hole table options.** To redefine additional hole table information, including a variety of default table, hole, and column properties, right-click on the hole table in the graphics window or browser, and select **Edit Hole Table...** from the pop-up menu. When you pick the **Edit Hole Table...** selection, the **Edit Hole Table** dialog box is displayed, with the **Options** tab shown. See Figure 18-41. This dialog box contains the same settings as the **Hole Table Style** area of the **Styles and Standards Editor** and can be used to override default hole table characteristics.

Figure 18-41.
The **Edit Hole Table** dialog box.





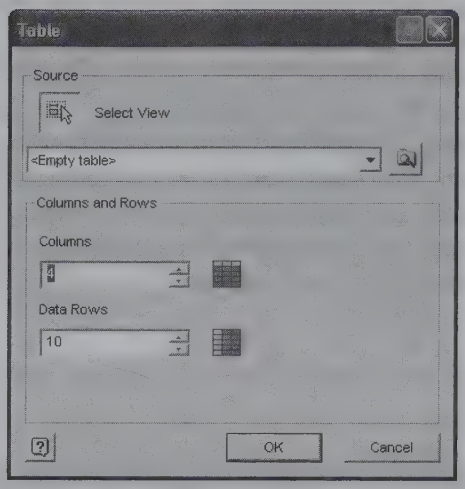
Exercise 18-12 Complete the Exercise on the Student CD.

Adding Tables

The primary function of the drawing **Table** tool is to reference iPart and iAssembly parameter tables for use in drawings. iParts and iAssemblies are described later in this text. However, the **Table** tool can be used to create tables not associated with iParts and iAssemblies. Parts lists, revision blocks, and tabular dimensioning tables are usually created using drawing annotation tools specifically designed for these applications. Most of these tools reference existing model or drawing data to produce tables. The **Table** tool can be used to form tables that are not parametrically controlled by model or drawing data, or for tables that cannot be made using other drawing annotation tools.

To access the **Table** tool and display the **Table** dialog box, shown in Figure 18-42, pick the **Table** button on the **Drawing Annotation Panel** bar; or pick the **Table** button

Figure 18-42.
The **Table** dialog box.



on the **Drawing Annotation Panel** toolbar. To create a table that references an iPart or iAssembly, first pick the iPart or iAssembly drawing view. The **Source** area will display the iPart or iAssembly file path, and the **Column and Rows** area will provide the **Column Chooser** button and the **Selected Columns** list box. These options allow you to define the content of the table based on an existing iPart or iAssembly parameter table. Once you choose columns, pick a location for the table on the sheet.

When creating a table that does not reference an iPart or iAssembly, the **Source** area will indicate that you are creating an <empty table>. Enter or pick the desired number of table columns and row using the **Columns** and **Data Rows** edit boxes. Once you define the number of columns and rows, pick a location for the table on the sheet. Once you create a table, you can edit much of the table information and position. A parts list can be moved like other Inventor items. When you select the parts list, green buttons are displayed at each corner. Use these buttons to move the parts list to a new location either in space or by linking a corner of the table to an existing drawing point, such as the corner of a border or title block. Most table editing options are controlled in the **Edit Table** dialog box, shown in Figure 18-43, accessed by double-clicking on the

Figure 18-43.
The **Edit Table** dialog box is used to add and modify table content.

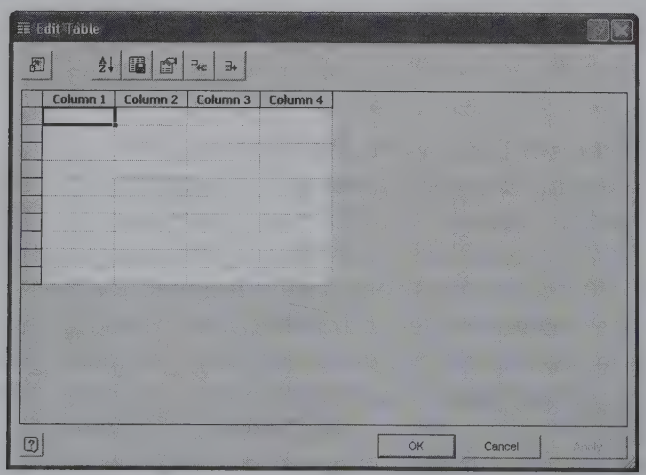


table or right-clicking on the table and selecting the **Edit** menu option. The following items are available in the **Edit Table** dialog box:

- **Column Chooser.** Select this button to open the **Table Column Chooser** dialog box, which is used to add and remove columns to and from the table. See Figure 18-44. The **Table Column Chooser** dialog box has a purpose similar to the **Property Chooser** and **Hole Table Column Chooser** dialog boxes described earlier, and allows you to add columns to the table by referencing parameter columns. When creating a blank table that does not reference a drawing view, nothing will be shown in the **Available Columns** list box while the **Selected Columns** list box displays default column names based on the number of columns you entered in the **Columns** edit box. To create a custom column, pick the **New Property** button, which opens the **Define New Property** dialog box. To remove a column from the table, select the column from the **Selected Columns** list box and pick the **Remove** button, or double-click on the column name. Columns in the **Available Columns** list box can be deleted using the **Delete** button, or added to the **Selected Columns** list box using the **Add** button. When two or more columns are specified, the columns are displayed in the table in the same order shown in the **Selected Columns** list box. To move a property up in the list, highlight the property and pick the **Move Up** button. Conversely, to move a property down in the list, highlight the property and select the **Move Down** button.
- **Sort.** When you select this button, the **Sort Table** dialog box is displayed, and allows you to modify the current table column order. See Figure 18-45. Select the column you want to sort first by picking a column from the **Sort by** drop-down list. Then specify whether you want the column displayed in ascending or descending order by selecting either the **Ascending** or **Descending** radio button. Next, if required, identify which column you want to sort second, when the first column is not available, by picking a column from the **Then by** drop-down list. Finally, if needed, identify which column you want to sort third, when the first and second columns are not available, by selecting a column from the second **Then by** drop-down list.
- **Export.** Pick this button if you want to export, or save, the parts list to an external, or non-Inventor type of file. When you select the **Export** button, the **Export Parts List** dialog box is displayed, and functions much like any other **Save As** dialog box. However, in addition you can enter the name of the table in the **Table Name** edit box.

Figure 18-44.
The **Table Column Chooser** dialog box.

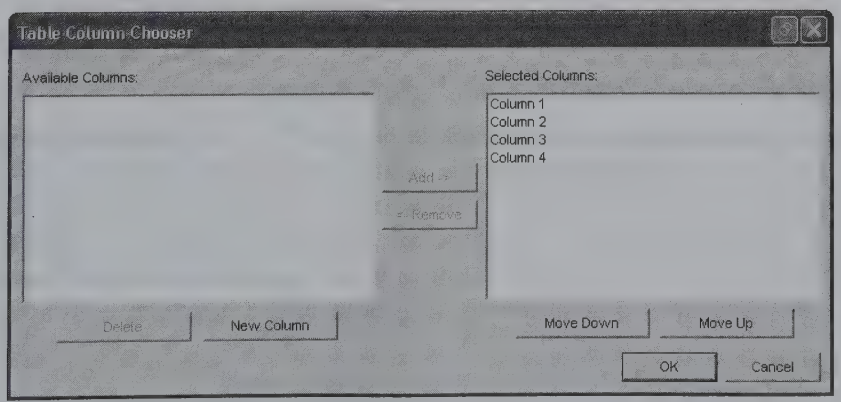
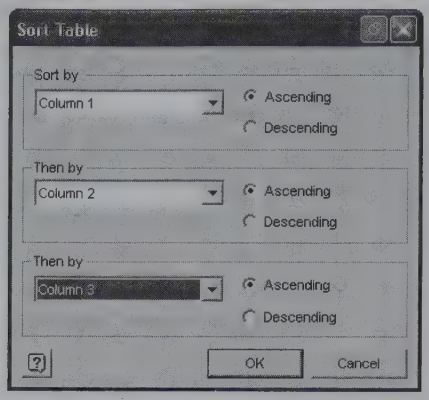


Figure 18-45.
The **Sort Table** dialog box.



- **Table Layout.** Choose this button to open the **Table Layout** dialog box, which allows you to override a variety of table layout characteristics. The options available in the **Table Layout** dialog box are the same as those in the **Table Style** area of the **Style and Standards Editor**. The only difference is that you cannot modify the heading and row gaps using the **Table Layout** dialog box.
- **Insert Row.** Pick this button to add a new row below the currently selected row.
- **Remove Row.** Pick this button to remove the selected row.
- **Table Spread Sheet.** The table spread sheet, or table, displays table information and allows you to modify table values. Many of the options associated with the table spread sheet have already been described. However, the spread sheet also contains many other table options. You can edit any of the information in the cells by selecting inside the cell, or by highlighting and redefining the existing value. Column width can also be edited by selecting the line in between two columns and dragging the column to the desired width. In addition, if you right-click on any of the spread sheet cells, the following menu options are listed:
  - **Visible.** When this option is selected, the row is visible. Conversely, deselect the **Visible** option to turn off, or hide, the row display.
  - **Wrap Table at Row.** Select this option to split, or divide, the column at the selected row. This option functions similar to the **Enable Automatic Wrap** check box in the **Table** dialog box. For large tables, you may want to split columns so that the comment is divided among a specified number of columns.



Exercise 18-13 Complete the Exercise on the Student CD.

Geometric Dimensioning and Tolerancing

Geometric dimensioning and tolerancing (*GD&T*) is the dimensioning and tolerancing of individual features of a part where the permissible variations relate to characteristics of form, profile, orientation, runout, or the location of features. GD&T uses symbols and related tolerances to display the desired specifications on the drawing. Inventor provides you with tools that allow you to generate feature control frames, feature identifier symbols, datum feature symbols, and datum target symbols.

Adding Feature Control Frames

A *feature control frame* may contain a geometric characteristic symbol, geometric tolerance, material condition symbol, and datum reference. A feature control frame is

divided into compartments containing the geometric characteristic symbol in the first compartment, followed by the geometric tolerance in a compartment and each datum reference in a separate compartment. A geometric tolerance is preceded by a diameter symbol if the geometric tolerance zone is cylindrical, and is generally placed with a related dimension on the drawing. See Figure 18-46. Feature control frames can be added to drawings using the **Feature Control Frame** tool. To access the **Feature Control Frame** tool, pick the **Feature Control Frame** button on the **Drawing Annotation Panel** bar; pick the **Feature Control Frame** button on the **Drawing Annotation Panel** toolbar; or press the [F] key on your keyboard.

Once you access the **Feature Control Frame** tool, use the cursor to select the feature, or a point, to which you want to add feature control frame information. Then pick a location for the feature control frame symbol. See Figure 18-47.

NOTE

If you do not want to create a leader from the feature control frame to the feature, do not pick the position of the leader shoulder. After you select the first point, press [Enter], or right-click and select the **Continue** menu option.

After you select the appropriate drawing geometry or a point on the sheet and locate the feature control frame, press the [Enter] key on your keyboard, or right-click and select the **Continue** menu option, to display the **Feature Control Frame** dialog box. See Figure 18-47. Use the following options in the **Feature Control Frame** dialog box to fully define a feature control frame:

- **Sym.** These buttons allow you to define the geometric characteristic symbols for the first, second, and third lines of feature control frame information. Select this button to access the symbols palette, and pick the desired symbol.
- **Tolerance.** These edit boxes are used to specify geometric tolerances. The **Tolerance 2** edit box is only available if you pick the **Allow Tolerance 2** check box in the **Feature Control Frame Style** area of the **Styles and Standards Editor**.

Figure 18-46.
Examples feature control frames created using Inventor.

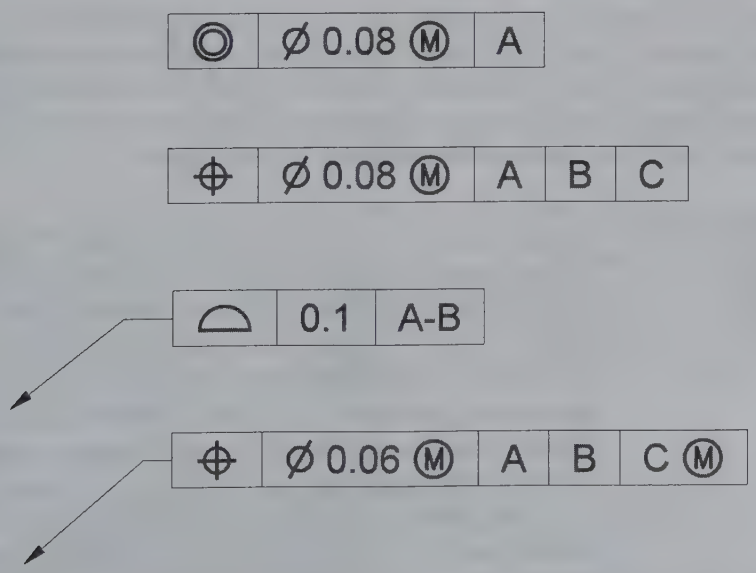
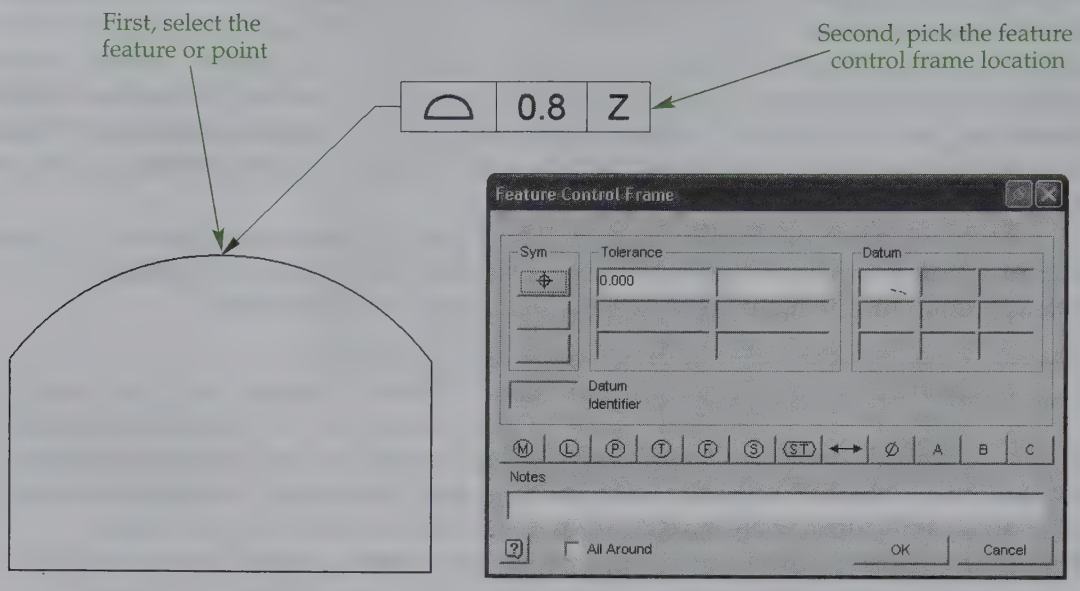


Figure 18-47.
Placing a feature control frame using the **Feature Control Frame** dialog box.



- **Datum.** These edit boxes allow you to enter primary, secondary, and tertiary datums in the feature control frame. The **Datum 2** edit box is available only when a value is entered in the **Datum 1** edit box, and the **Datum 3** edit box is available only when a value is entered in the **Datum 2** edit box.
- **Datum Identifier.** Use this edit box to reference a feature identifier symbol when used. The **Datum Identifier** edit box is only available if you pick the **Datum ID** check box in the **Feature Control Frame Style** area of the **Styles and Standards Editor**.
- **Insert Modifier.** These buttons control multiple geometric dimensioning and tolerancing information, such as material condition and tolerance zone descriptors. To use these buttons, pick a location inside the desired edit box, and choose the required symbol button. The symbol should appear in the selected edit box.
- **Notes.** Use this edit box to insert a note below the feature control frame.
- **All Around.** Select this check box to add an all-around symbol on the leader, next to the feature control frame.

When you fully define the feature control frame specifications in the **Feature Control Frame** dialog box, pick the **OK** button to create the feature control frame. You will notice the **Feature Control Frame** tool is still active. Continue placing feature control frames as required or right-click and select the **Done** menu option, press the [Esc] key on your keyboard, or access a different tool, to exit the tool.

NOTE



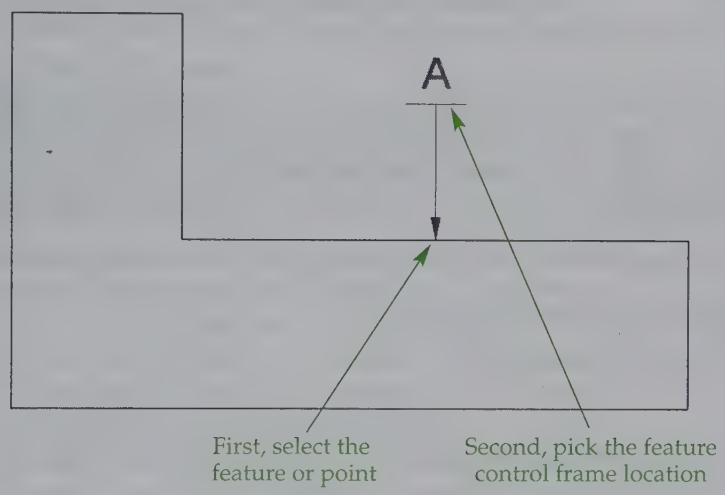
Once inserted, you can modify feature control frame information by right-clicking on the feature control frame and selecting the **Edit Feature Control Frame...** pop-up menu option. You can also override the units specified in the **Feature Control Frame Style** area of the **Styles and Standards Editor** for a specific feature control frame by right-clicking on the feature control frame and selecting the **Edit Unit Attributes** pop-up menu option. You must deselect the **Use Standards Notation** check box in the **Edit Unit Attributes** dialog box to override the default unit settings.

Placing Feature Identifier Symbols

The *feature identifier* symbol is not used in the ASME standard. It is a symbol that is used to correlate certain features to specifications in the ISO, DIN, and JIS standards. See Figure 18-48. Feature identifier symbols are placed using the **Feature Identifier Symbol** tool. To access the **Feature Identifier Symbol** tool, pick the **Feature Identifier Symbol** button on the **Drawing Annotation Panel** bar, or pick the **Feature Identifier Symbol** button on the **Drawing Annotation Panel** toolbar.

Once you access the **Feature Identifier Symbol** tool, use the cursor to select the feature, or a point, to which you want to add a feature identifier symbol, or pick a point in the drawing sheet. Then pick a location for the symbol to specify the leader length. See Figure 18-48. After you select appropriate drawing geometry or a point on the drawing sheet, and locate the feature identifier symbol, press the [Enter] key on your keyboard, or right-click and select the **Continue** menu option to display the **Format Text** dialog box. The default feature identifier value is displayed in the **Format Text** dialog box and can be modified.

Figure 18-48.
Placing a feature identifier symbol.



NOTE

If you select two points, in addition to the initial feature or point, to define the location of the feature identifier symbol, the **Format Text** dialog box automatically opens.

When you are satisfied with the value of the feature identifier symbol and finish defining options in the **Format Text** dialog box, pick the **OK** button to generate the feature identifier symbol. The **Feature Identifier Symbol** tool will still be active, allowing you to continue placing feature identifier symbols. To exit the **Feature Identifier Symbol** tool, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, or access a different tool.

Placing Datum Feature Symbols

A *datum feature symbol* is also referred to as a *datum identification symbol*, in Inventor. *Datums* are considered theoretically perfect surfaces, planes, points, or axes, and are identified by datum feature symbols on the drawing. The datum feature

symbol is a square box that contains the datum reference letter. The box is connected to a triangular datum arrow that contacts the datum surface or related application. Datum feature symbols are attached to the surface with a leader or off an extension line, with a diameter or center plane dimension, or attached to a feature control frame. See Figure 18-49. Datum feature symbols are placed using the **Datum Identifier Symbol** tool. To access the **Datum Identifier Symbol** tool, pick the **Datum Identifier Symbol** button on the **Drawing Annotation Panel** bar, or pick the **Datum Identifier Symbol** button on the **Drawing Annotation Panel** toolbar.

Once you access the **Datum Identifier Symbol** tool, use your cursor to select a feature or a point to which you want to add a datum identifier symbol, or pick a point in the drawing sheet. Then pick a location for the symbol to specify the leader length. See Figure 18-50. After you select appropriate drawing geometry or a point on the drawing sheet, and locate the datum identifier symbol, press the [Enter] key on your keyboard, or right-click and select the **Continue** menu option to display the **Format Text** dialog box, described earlier in this chapter. The default datum identifier value is displayed in the **Format Text** dialog box and can be modified.

NOTE

If you select two points, in addition to the *initial* feature or point, to define the location of the datum feature symbol, the **Format Text** dialog box automatically opens.

When you are satisfied with the value of the datum identifier symbol and finish defining options in the **Format Text** dialog box, pick the **OK** button to generate the datum identifier symbol. The **Datum Identifier Symbol** tool will still be active, allowing you to continue placing datum identifier symbols. To exit the **Datum Identifier Symbol**

Figure 18-49.
Examples of various datum feature symbols.

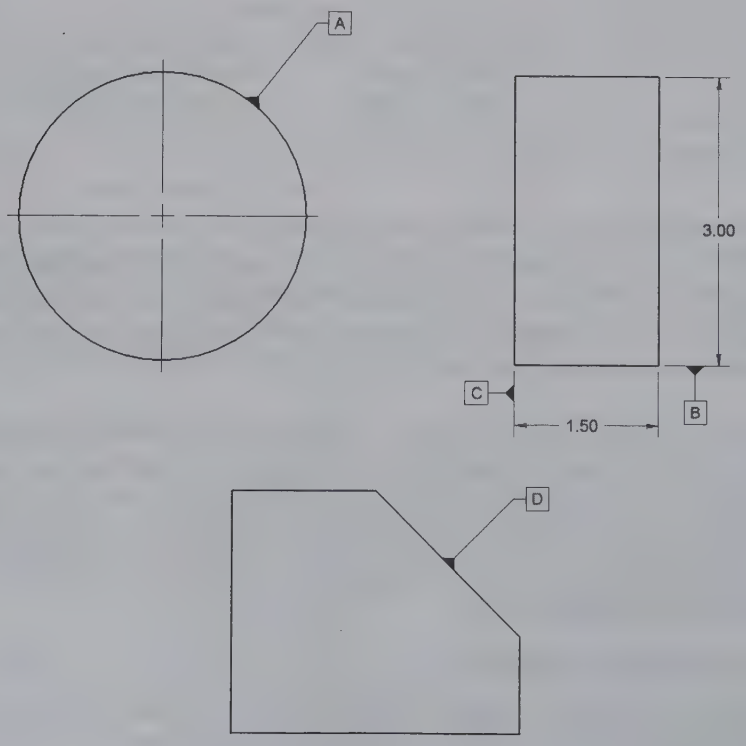
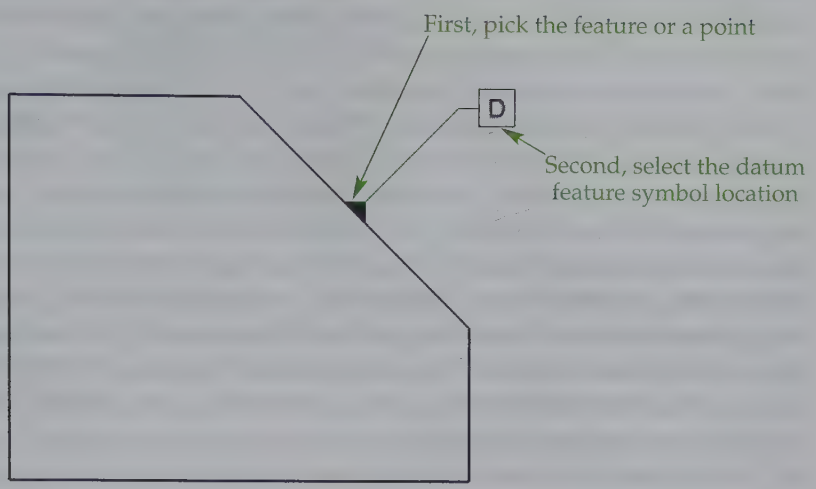


Figure 18-50.
Placing a datum identifier symbol.



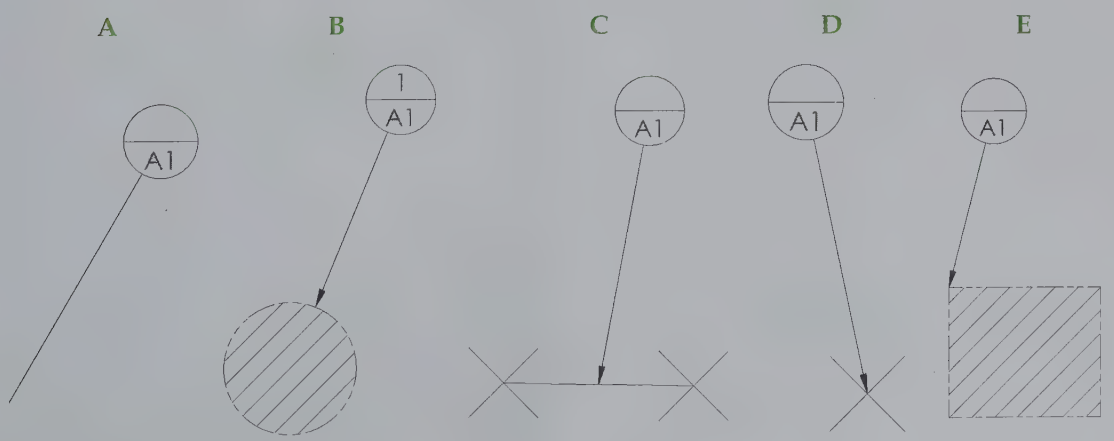
tool, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, or access a different tool.

Adding Datum Target Symbols

Datum targets are used to specify points, lines, or areas of contact on a part that establishes datums when it is not possible or appropriate to use a surface. A datum target symbol is drawn as a circle with a horizontal line through the center. The top half of the circle is left blank, unless the datum target symbol refers to a circular target area. In this case, the size of the datum target area is provided in the top half of the datum target symbol. The bottom half of the datum target symbol is used to identify the related datum with the datum reference letter and datum target number assigned sequentially to the datum, such as A1, A2, and A3. A datum target symbol is connected to the related datum target point, line, or area with a leader that typically does not have an arrowhead at the end. See Figure 18-51.

Inventor provides five separate tools, **Datum Target - Leader**, **Datum Target - Circle**, **Datum Target - Line**, **Datum Target - Point**, and **Datum Target - Rectangle** for

Figure 18-51.
Examples of various types of datum targets.



adding datum target symbols for various applications. The **Datum Target - Leader** tool allows you to place a datum target with a leader attached to the specified feature or point, as shown in Figure 18-51A. The **Datum Target - Circle** tool allows you to place a datum target with a leader attached to a circular datum target area, as shown in Figure 18-51B. The **Datum Target - Line** tool allows you to place a datum target with a leader attached to a specified line, with points at the line ends, as shown in Figure 18-51C. The **Datum Target - Point** tool allows you to place a datum target with a leader attached to a point, as shown in Figure 18-51D. The **Datum Target - Rectangle** tool allows you to place a datum target with a leader attached to a rectangular datum target area, as shown in Figure 18-51E. Each of the different types of datum targets are contained in a flyout button in the **Drawing Annotation Panel** toolbar and panel bar, or the **Drawing Datum** toolbar, and can be accessed by picking the desired datum target tool button from the **Datum Target** flyout button on the **Drawing Annotation Panel** bar; picking the desired datum target tool button from the **Datum Target** flyout button on the **Drawing Annotation Panel** toolbar; or selecting the desired datum target tool button from the **Drawing Datum** toolbar.

Once you access one of the available datum target tools, use the following information to create the datum target:

- **Datum Target - Leader.** First, use your cursor to select the feature, or a point, to which you want to add a datum target, or pick a point on the drawing sheet. Then, pick a location for the datum target symbol to specify the leader length. See Figure 18-52. After you select appropriate drawing geometry or a point on the drawing sheet, and locate the datum target symbol, press the [Enter] key on your keyboard, or right-click and select the **Continue** menu option to display the **Datum Target** dialog box. See Figure 18-53. A dimension can be entered in the **Dimension** edit box, but is typically not used for a leader application. Specify the datum in the **Datum** edit box. Then pick the **OK** button to generate the datum target. The **Datum Target - Leader** tool will still be active, allowing you to continue placing leader datum targets. To exit the **Datum Target - Leader** tool, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, or access a different tool.

Figure 18-52.
Placing a leader datum target.

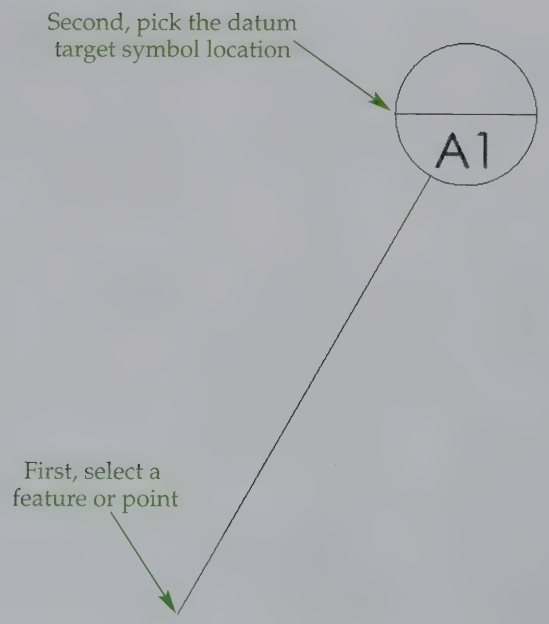
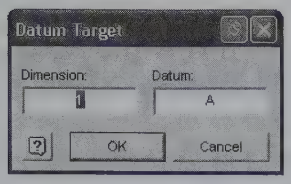


Figure 18-53.
The **Datum Target** dialog box.



- **Datum Target - Circle.** First, use your cursor to select the center point of the circular datum target area. Then pull the datum target area circle edge to the desired size, and pick the position to specify the circle diameter. Next, pick a location for the datum target symbol to specify the leader length. See Figure 18-54. After you create the circular area and locate the datum target symbol, press [Enter], or right-click and select the **Continue** menu option to display the **Datum Target** dialog box, shown in Figure 18-53. Enter the diameter of the area in the **Dimension** edit box, and the datum in the **Datum** edit box. Then pick the **OK** button to generate the datum target. The **Datum Target - Circle** tool will still be active, allowing you to continue placing circular area datum targets. To exit the **Datum Target - Circle** tool, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, or access a different tool.
- **Datum Target - Line.** First, use your cursor to select the start point of the datum target line. Then pull the datum target line to the desired location, and pick the position to specify the length of the line. Next, pick a location for the datum target symbol to specify the leader length. See Figure 18-55. After you create the line and locate the datum target symbol, press the [Enter] key on your keyboard, or right-click and select the **Continue** menu option to display the **Datum Target**

Figure 18-54.
Placing a circle datum target.

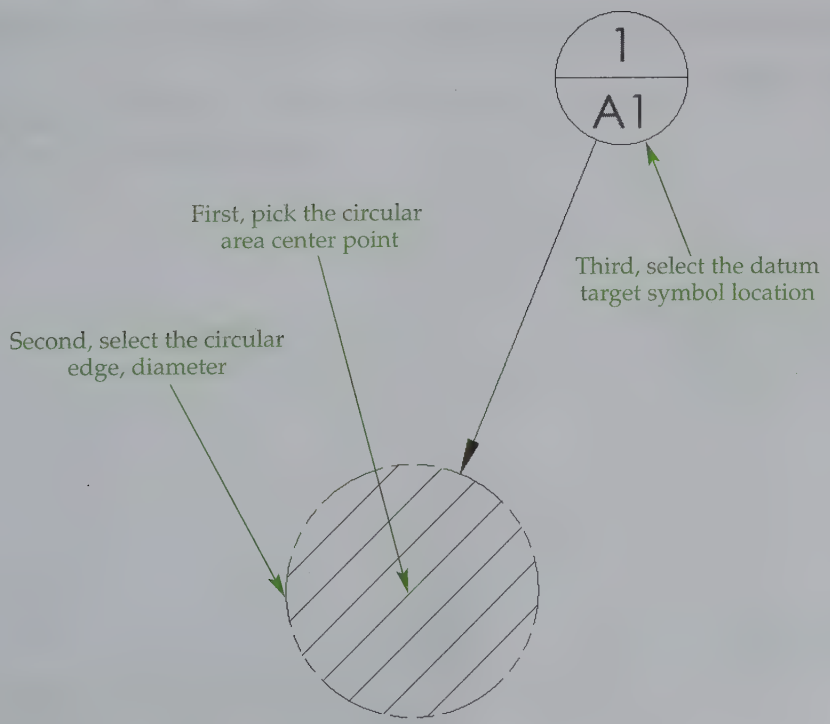
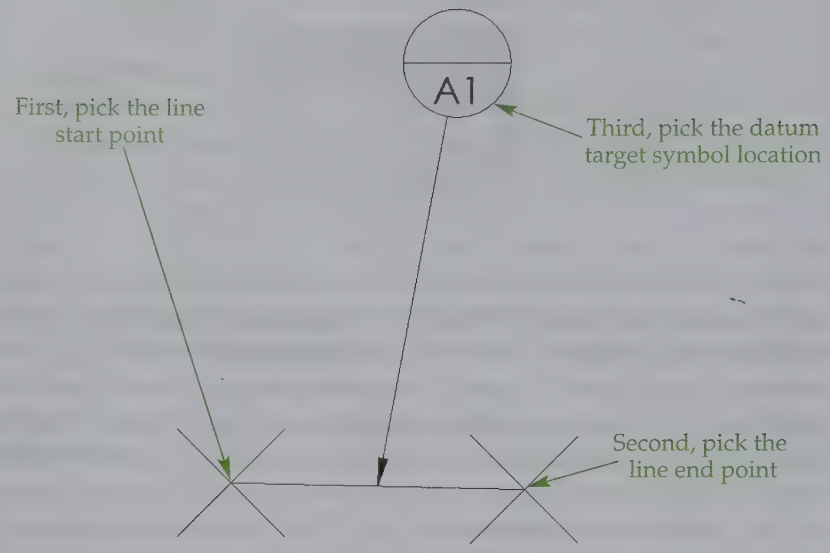


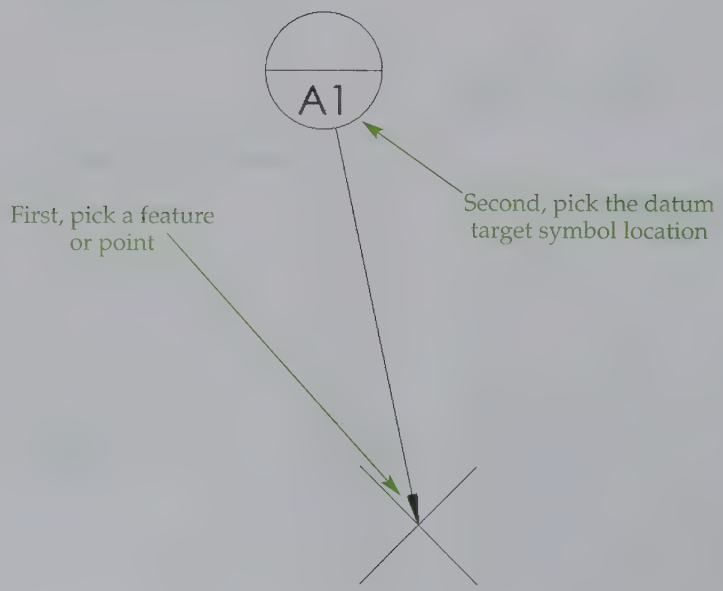
Figure 18-55.
Placing a line datum target.



dialog box, shown in Figure 18-53. A dimension can be entered in the **Dimension** edit box, but is typically not used for a line application. Specify the datum in the **Datum** edit box. Then pick the **OK** button to generate the datum target. The **Datum Target - Line** tool will still be active, allowing you to continue placing line referencing datum targets. To exit the **Datum Target - Line** tool, right-click and select the **Done** menu option, press [Esc], or access a different tool.

- **Datum Target - Point.** First, use the cursor to select the feature, or point, to which you want to add a datum target, or pick a point on the drawing sheet. Then, pick a location for the datum target symbol to specify the leader length. See Figure 18-56. After you select appropriate drawing geometry or a point on the drawing sheet, and locate the datum target symbol, press the [Enter] key on your keyboard, or right-click and select the **Continue** menu option to display the

Figure 18-56.
Placing a point datum target.



Datum Target dialog box, shown in Figure 18-53. A dimension can be entered in the **Dimension** edit box, but is typically not used for a point application. Specify the datum in the **Datum** edit box. Then pick the **OK** button to generate the datum target. The **Datum Target - Point** tool will still be active, allowing you to continue placing point referencing datum targets. To exit the **Datum Target - Point** tool, right-click and select the **Done** menu option, press [Esc], or access a different tool.

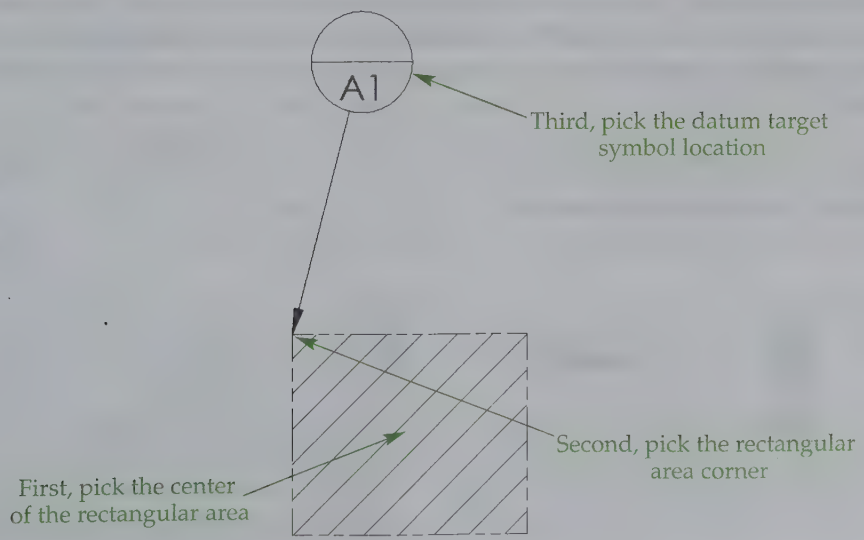
- **Datum Target - Rectangle.** First, use the cursor to select the center point of the rectangular datum target area. Then pull the datum target area rectangle corner to the desired width and length, and pick the position to specify the rectangle size. Next, pick a location for the datum target symbol to specify the leader length. See Figure 18-57. After you create the rectangular area and locate the datum target symbol, press the [Enter] key on your keyboard, or right-click and select the **Continue** menu option to display the **Datum Target** dialog box, shown in Figure 18-53. A dimension can be entered in the **Dimension** edit box, but is typically not used for a rectangle application. Specify the datum in the **Datum** edit box. Then pick the **OK** button to generate the datum target. The **Datum Target - Rectangle** tool will still be active, allowing you to continue placing rectangular-area datum targets. To exit the **Datum Target - Rectangle** tool, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, or access a different tool.

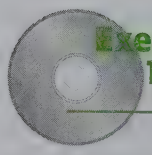
NOTE



Once inserted, you can modify datum target symbol information by right-clicking on the datum target symbol and selecting the **Edit Datum Target...** pop-up menu option. You can also attach another datum target symbol to the existing symbol by right-clicking on the datum target symbol and selecting the **Attach Balloon** pop-up menu option. In addition, you can override the units specified in the **Datum Target Style** area of the **Styles and Standards Editor** for a specific datum target symbol by right-clicking on the datum target symbol and selecting the **Edit Unit Attributes** pop-up menu option. You must deselect the **Use Standards Notation** check box in the **Edit Unit Attributes** dialog box to override the default unit settings.

Figure 18-57.
Placing a rectangle datum target.





Complete the Exercise on the Student CD.

Custom Drawing Symbols

The default symbols created using Inventor annotation tools are effective for most applications. For example, typically there is no reason to develop your own datum target symbols, because these symbols are created easily and effectively using one of the datum target symbol tools. Still, at some point during drawing creation you may need to add your own custom symbols. For example, your drawings may require company or industry specific symbols that are not available from standard Inventor tools. See Figure 18-58.

Just as with custom borders, title blocks, and draft views, custom symbols are created using sketch tools. To develop a custom symbol, select the **Define New Symbol** option from the **Format** pull-down menu, or right-click on the **Sketched Symbols** folder of the **Drawing Resources** folder in the browser, and pick the **Define New Symbol** pop-up menu option. When you pick the **Define New Symbol** option, the panel bar automatically enters **Drawing Sketch Panel** mode and you can access the **Drawing Sketch Panel** toolbar. The drawing sketch environment is exactly the same when sketching a symbol as when sketching a custom border or title block. For more information regarding sketching in the drawing environment, refer to previous chapters.

NOTE



Use the **Insert AutoCAD file** tool in the sketch environment to create a symbol from an existing AutoCAD file.

After you sketch your custom symbol using sketch tools, you are ready to exit the sketch environment and generate the symbol by right-clicking and selecting the **Save Sketched Symbol** pop-up menu option; picking the **Return** button on the **Inventor Standard** toolbar; or selecting the **Save Sketched Symbol** option from the **Format** pull-down menu.

When you exit the sketch environment, the **Sketched Symbol** dialog box is displayed, allowing you to save the symbol. See Figure 18-59A. Type the symbol name in the **Name** edit box, and pick the **Save** button to create the symbol. If you pick the **Discard** button, the sketch is deleted, and the symbol does not generate. The symbol you create as a sketch is transformed into an insertable feature. The new insertable symbol is contained in the **Sketched Symbols** folder of the **Drawing Resources** folder in the browser, along with other existing drawing symbols. See Figure 18-59B.

Figure 18-58.
Examples of custom drawing symbols.

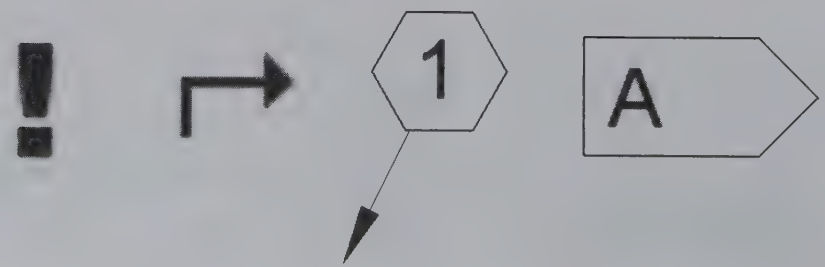
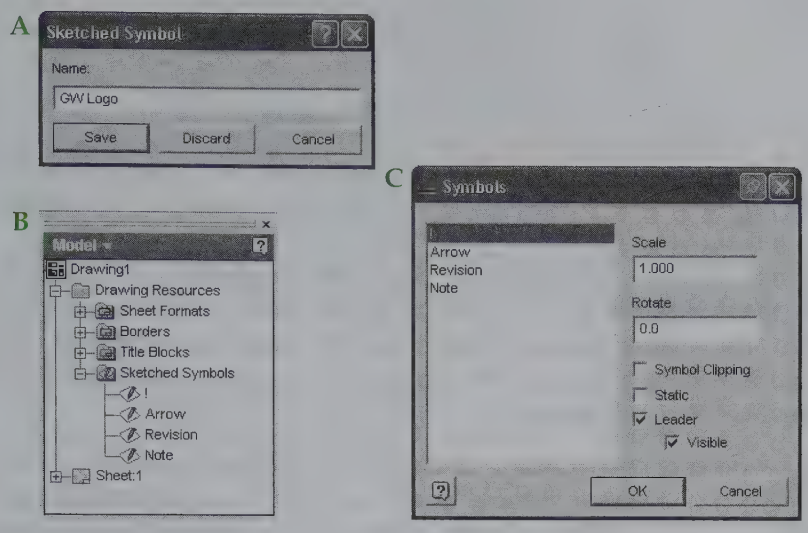


Figure 18-59.
A—The **Sketched Symbol** dialog box. B—Symbols contained in the **Sketched Symbols** folder. C—The **Symbols** dialog box.



There are a few different methods of inserting symbols onto a drawing sheet. One option is to either double-click the symbol name in the browser, or right-click on the symbol name in the browser and select the **Insert** menu option. Next, pick a location for the first symbol on the sheet. You can continue placing multiple copies of the same symbol, if required, by selecting additional locations on the drawing sheet. Then right-click and select the **Done** pop-up menu option, or press the [Esc] key on your keyboard to exit the tool. Another method for placing custom symbols is to use the **Symbols** tool. In addition to inserting the symbol, the **Symbols** tool allows you to scale, rotate, and add a leader, if required, to the symbol. To access the **Symbols** tool, pick the **Symbols** button on the **Drawing Annotation Panel** bar, or pick the **Symbols** button on the **Drawing Annotation Panel** toolbar.

The **Symbols** dialog box is displayed when you access the **Symbols** tool. See Figure 18-59C. The **Symbols** dialog box contains all the custom symbols currently available in the drawing, corresponding to the symbols listed in the **Sketched Symbols** folder of the **Drawing Resources** folder in the browser. To use the **Symbols** dialog box, first select the symbol you want to insert from the **Symbols** list box. Then, if required, pick the **Scale** button to activate the **Scale** edit box, and enter the desired scale. The scale you enter will increase or decrease the size of the symbol in reference to its original size. For example, a scale factor of 2 will increase the size of a ∅2" circular shaped symbol to ∅4". The next step, if necessary, is to define an insertion angle by picking the **Rotate** button, and entering or selecting the desired rotation angle in the **Rotate** edit box. The following check boxes are also available in the **Symbols** dialog box:

- **Symbol Clipping.** Choose this check box to trim dimensions away from the symbol, when the symbol is inserted on dimension objects. Or, clear the check box to place the symbol directly over dimensions without clipping the dimensions.
- **Static.** With this check box deselected, symbols can be rotated, resized, and moved. With the **Static** check box selected, symbols cannot be rotated or resized, but can still be moved.
- **Leader.** Pick this check box to automatically add a leader to the symbol. Toggle the visibility of the leader by selecting or deselecting the **Visibility** check box. With the **Visibility** check box deselected, the leader line appears dashed when you initially place and move the symbol, but is hidden at all other times.

Once you finish specifying custom symbol insertion options, pick the **OK** button to exit the **Symbols** dialog box.

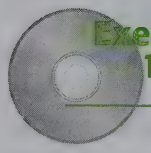
If you selected the **Leader** check box in the **Symbols** dialog box, the first step to place the symbol is to select the feature or point at which you want to start the leader and add the symbol, or pick a point on the drawing sheet. Then drag the leader and symbol to the desired location and pick the position. See **Figure 18-60.** Next, press the [Enter] key on your keyboard, or right-click and select the **Continue** menu option to generate the first symbol and leader. You can continue placing multiple copies of the same symbol, if required, by selecting additional leader and symbol locations on the drawing sheet. When finished, right-click and select the **Done** pop-up menu option, or press the [Esc] key on your keyboard to exit the tool.

If you did not select the **Leader** check box in the **Symbols** dialog box, all you have to do to place the symbol is pick a location on the sheet. You can continue placing multiple copies of the same symbol, if required, by selecting additional points on the drawing. Then, right-click and select the **Done** pop-up menu option, or press the [Esc] key on your keyboard to exit the tool.

NOTE

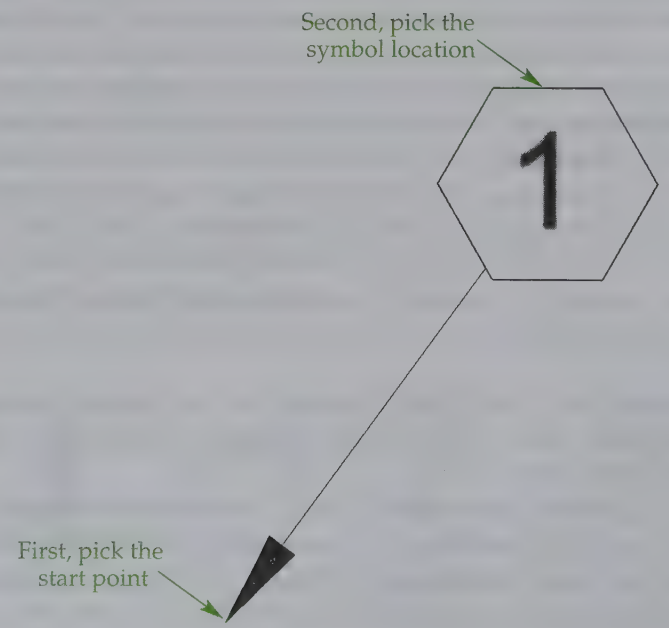


Any symbol inserted using the **Symbols** dialog box, with the **Static** check box deselected, can be rotated by highlighting the symbol and dragging the blue dot to the desired angle, and scaled by highlighting the symbol and dragging one of the yellow dots to the desired size. Green dots are available even if the **Static** check box is selected, and can be used to relocate the leader origin and symbol.



Exercise 18-15 Complete the Exercise on the Student CD.

Figure 18-60.
Placing a custom symbol with a leader.



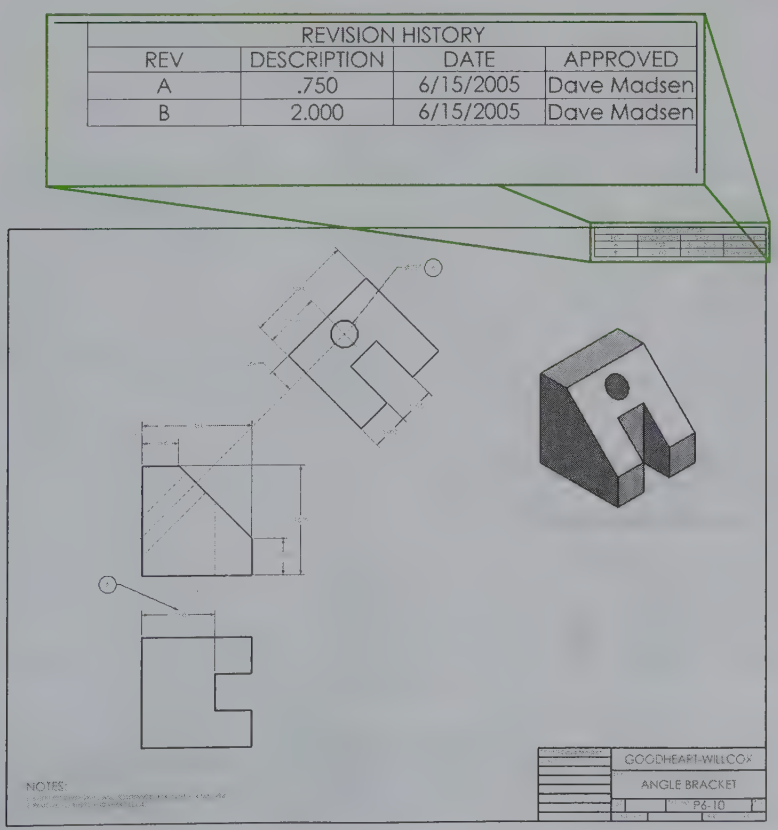
Adding a Revision Table and Revision Tags

Often, part models and drawings are altered several times throughout the life of a product in order to remanufacture the part based on changes. Typically part modifications are documented on drawings using revision history blocks and revision symbols. *A **revision history block**,* also known as a ***revision table*** or ***revision block***, records drawing changes, and is usually placed in the upper-right corner of the drawing. Revision tables are created using the **Revision Table** tool. A ***revision symbol***, also known as a ***revision tag***, identifies the location at which the engineering change occurs, and corresponds to a specific entry in the revision table. Revision tags are created using the **Revision Tag** tool. Figure 18-61 shows an example of a drawing revision.

Though you may develop your own methods for adding revision information to a drawing, the following approach is typically most effective for adding a revision table and revision tags:

- First Revision:
  1. Place a revision table.
  2. Edit the revision table properties and entries as required.
  3. Place revision tags that correspond to the first revision, and first and only revision table row.
- Second Revision:
  1. Add a row to the revision table that corresponds to the second revision.
  2. Edit the revision table properties and second row entries as required.
  3. Place revision tags that correspond to the second revision, and second revision table row.
- Continue the second revision process for each additional revision.

Figure 18-61.
An example of a drawing revision, complete with a revision table and revision symbols.



To place a revision table, access the **Revision Table** tool by picking the **Revision Table** button from the flyout button on the **Drawing Annotation Panel** bar; picking the **Revision Table** button from the flyout button on the **Drawing Annotation Panel** toolbar; or picking the **Revision Table** button on the **Drawing Revision Table** toolbar.

Once you access the **Revision Table** tool, pick a location on the drawing sheet where you want the revision table to be positioned. Automatic coincident constraints should allow you to place the revision table in reference to the border, without having to select a border corner. After you generate the default revision table using the **Revision Table** tool, there are a number of options for modifying and adding information depending on the application and desired table appearance. See Figure 18-62. The following revision table items can be redefined using the techniques described:

- **Resize rows and columns.** A revision table *row* is the horizontal area that contains information about each revision—every revision has its own row. A revision table *column* is the vertical area that contains a specific set of revision entries, such as Zone. You can expand and contract rows and columns by moving your cursor over a row or column line until you see the resize icon, and then drag the row or column to the desired location. See Figure 18-63.
- **Add and remove revision table rows.** To add an additional revision to the revision table without deleting and recreating the table, right-click on the revision table in the graphics window or browser, and select the **Add Row** pop-up menu option. Conversely, you can remove a row from the table by right-clicking on the row you want to remove from the revision table, and selecting the **Delete Row** pop-up menu option.
- **Edit revision table options.** To redefine additional revision table information, including a variety of default table, revision, and column properties, right-click on the revision table in the graphics window or browser, and select the **Edit** pop-up menu option to access the **Edit Revision Table** dialog box. See Figure 18-64. The **Edit Revision Table** dialog box contains the following areas:

Figure 18-62.
Manipulating revision table row and column sizes.

Original

REVISION HISTORY				
ZONE	REV	DESCRIPTION	DATE	APPROVED
1	A	Value	6/15/2005	Dave Madsen

REVISIONS				
ZONE	REV	DESCRIPTION	DATE	APPROVED
1	A	Value	6/15/2005	Dave Madsen

After changing the row and column sizes

Figure 18-63.
Modifying revision table row and column sizes.

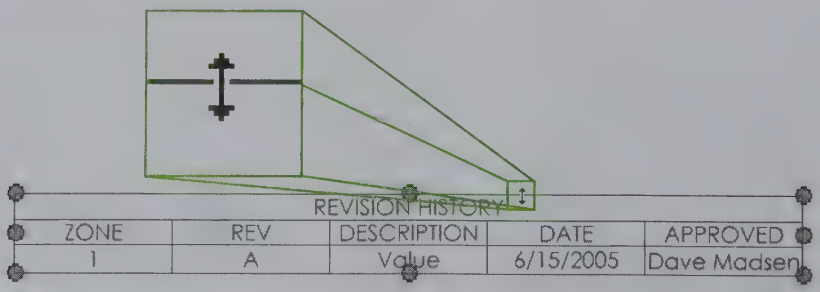
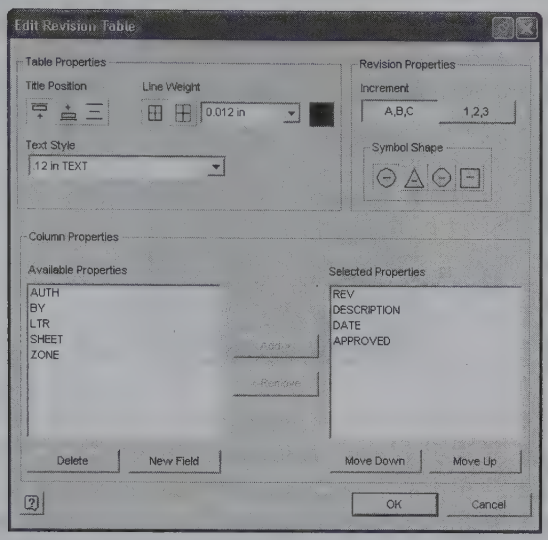
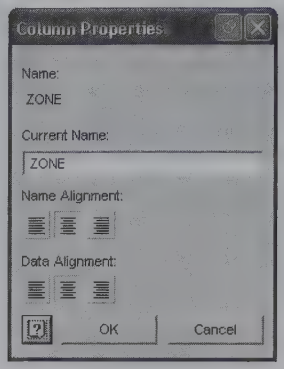


Figure 18-64.
The **Edit Revision Table** dialog box.



- **Table Properties.** This area allows you to modify various default revision table settings not related to revision tags or revision table columns. By default, the **Top** title position button is selected, which means that the revision table title is located at the top of the table. Pick the **Bottom** title position button to move the title to the bottom of the table, or pick the **None** title position button to completely remove the revision table title. To change the line weight and color of the revision table border, pick the **Outside** button. Then, select the desired line weight from the **Line Weight** drop-down list, and pick the **Color** button to access the **Color** dialog box, which allows you to change the revision table border color. To change the line weight and color of cell borders, which are the lines that separate table entries and titles, pick the **Inside** button. Then, select the desired line weight from the **Line Weight** drop-down list, and pick the **Color** button to access the **Color** dialog box, which allows you to change the revision cell border color.
- **Revision Properties.** Use this area to define various revision table and revision tag display characteristics. Pick the **A,B,C Increment** button to use letters for revision tags, and for REV table entries. Pick the **1,2,3 Increment** button to use numbers for revision tags, and for REV table entries. Then, specify the type of revision symbol to use in the drawing by selecting the desired shape from the **Symbol Shape** area.
- **Column Properties.** Use this area to add, remove, relocate, and rename revision table column properties. Column properties that are currently displayed in the revision table are shown in the **Selected Properties** list box. Columns listed from top to bottom in the **Selected Properties** list box are displayed from left to right in the revision table. You can rearrange the revision table columns by selecting the column you want to move, and picking the **Move Down** and **Move Up** buttons. To redefine a column name and alignment in the cell, right-click on the desired property name in the **Selected Properties** list box and select the **Edit Column Properties** pop-up menu option. This opens the **Column Properties** dialog box for the selected property. See Figure 18-65. You can rename the column by entering a new name in the **Current Name** edit box. Modify the alignment of the column name by selecting the desired **Name Alignment** button. You can also redefine

Figure 18-65.
The **Column Properties** dialog box.



the alignment of the column cells, or column data, by selecting the desired **Data Alignment** button. The **Column Properties** area also allows you to add and remove revision table columns. To remove an existing column from the table, pick the column name from the **Selected Properties** list box, and choose the **Remove** button. Conversely, to add another column to the table, pick the column name from the **Available Properties** list box, and choose the **Add** button. To completely remove a property from the **Column Properties** area, pick the property in the **Available Properties** list box, and pick the **Delete** button. Finally, you can create your own custom property to include in the revision table by selecting the **New Field** button. When you pick **New Field**, the **Define New Field** dialog box is displayed, allowing you to enter the new column name. See Figure 18-66.

When you finish redefining revision table properties using the **Revision Table Options** dialog box, pick the **OK** button to accept the changes and exit the dialog box.

- Redefine table entries. You can modify the values in any of the table cells by double-clicking on the existing value to access the **Format Text** dialog box, described earlier.

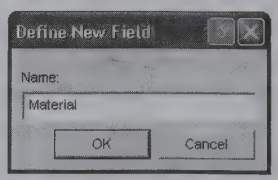
NOTE



Though you can modify any table cell values using the **Format Text** dialog box, you should only use the **Format Text** dialog box to edit values that cannot be redefined in the **Edit Revision Table** dialog box.

To place revision tags, access the **Revision Tag** tool by picking the **Revision Tag** button from the flyout button on the **Drawing Annotation Panel** bar; picking the **Revision Tag** button from the flyout button on the **Drawing Annotation Panel** toolbar; or picking the **Revision Tag** button on the **Drawing Revision Table** toolbar.

Figure 18-66.
The **Define New Field** dialog box.



Once you access the **Revision Tag** tool, select a feature, a point on drawing geometry, or pick a point anywhere on the drawing sheet. Then, if you want to use a leader, pull the leader shoulder to the desired location and pick the point. Once you locate the tag leader shoulder, press the [Enter] key on your keyboard, or right-click and select the **Continue** menu option to generate the tag. If you do not want to use a leader, press the [Enter] key on your keyboard, or right-click and select the **Continue** menu option immediately after you pick the first point. You can continue placing multiple copies of the same revision tag, if required, by repeating the selection process. Then, right-click and select the **Done** pop-up menu option, or press the [Esc] key on your keyboard to exit the tool.

NOTE



Revision tag characteristics are specified in the **Edit Revision Table** dialog box.



Exercise 18-16 Complete the Exercise on the Student CD.

Modifying Dimensions

Dimensions can be modified by changing display characteristics or overriding specific dimension style variables after dimensions have been created. For most dimensions, the dimension style you establish and use according to the active standard should be appropriate for most applications. In addition, to change specific object values you should modify the actual part, not just drawing dimension values, to preserve the parametric associations of drawing dimension values. However, there are times when you must change the display of dimensions or add data to dimension values. The following list represents several options for modifying existing dimensions:

- Stretch extension lines to offset a dimension farther from an object, or create extension lines that are not perpendicular to the object.
- Move a leader arrowhead, or stretch a leader shoulder.
- Move dimension text to a new location.
- Extend or shorten centerlines and center marks.
- Change dimension value specifications, such as the default tolerance.
- Edit dimension value information, such as the parameters in a feature control frame.

Dimension modification options are specific to the dimensions you create. For example, you can choose to reverse the direction of ordinate dimensions, or change the default leader arrowhead. Overall, each section of this chapter explores possible dimension modification and override techniques related to certain dimensions and annotations. Still, there are some general editing items with which you may want to become familiar.

To modify physical dimension properties, such as a dimension line offset, pick the dimension you want to change. You will notice several green dots, which designate movable dimension items. See Figure 18-67. Once you see the **Move** icon, pick one of the points or the dimension text and drag the dimension information to the desired location.

If you move the cursor over the green dot and the **Arrow** icon is displayed, double-click the dot to open the dialog box for the **Change** specific dimension feature. For example, if you double-click on a leader arrowhead, the **Arrowhead** dialog box is displayed, which

allows you to override the arrowhead style. You can also modify, or override, dimension characteristics by right-clicking on a dimension and selecting a menu option. For example, Figure 18-68 shows the menu options available when you right-click on a horizontal linear dimension, created using the **Baseline Dimension** tool.

Figure 18-67.
Examples of dimension alteration points.

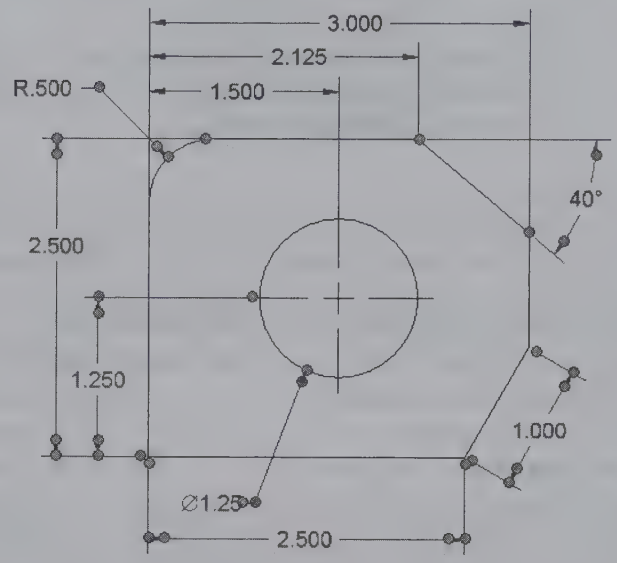
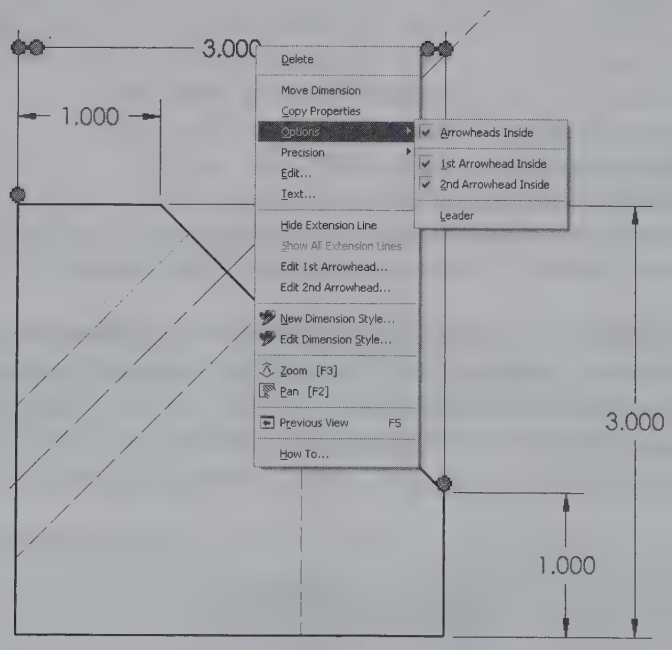
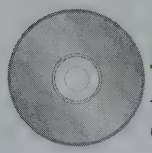


Figure 18-68.
Right-clicking on the **3.000** dimension reveals these menu options.





Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. Discuss the use of dimensions.
2. Explain why most Inventor drawing files contain both *model* and *drawing* dimensions.
3. Describe *model dimensions.*
4. Define *drawing dimensions.*
5. Briefly discuss the parametric nature of model dimensions.
6. Identify and give an example of the *model dimensions* that can be shown in a view.
7. Explain the function and use of *drawing annotation tools.*
8. Briefly discuss the function of the **General Dimension** tool.
9. What are *linear dimensions*?
10. Define *aligned dimensions* and give the Inventor description of aligned dimensions.
11. Describe *radial dimensions.*
12. Explain where *diameter dimensions* are used and identify the symbol used with diameter dimensions.
13. Identify where *angular dimensions* are used.
14. What does Inventor call *datum dimensions*?
15. Describe the use of datum dimensions.
16. Why are adjacent dimensions staggered when doing datum dimensioning?
17. Define specific notes and give another name used to identify specific notes.
18. What are general notes?
19. Name and identify the line that is usually used to connect a specific note to a feature.
20. Give another name for *ordinate dimensioning* and explain the purpose of ordinate dimensioning.
21. Explain the use of ordinate dimensioning in a traditional manner and with the use of a table.
22. Describe *tabular dimensioning.*
23. What is a *hole table row*?
24. Identify a *hole table column.*
25. In general, how are hole and thread notes typically dimensioned?
26. Explain how Unified and American National Thread notes are formatted.
27. Show how metric thread notes are formatted.
28. Define *surface finish* and describe the surface finish symbol.
29. Give a general definition of GD&T.
30. What is a *feature control frame,* explain what it looks like, and where it is placed on a drawing?
31. What is a *feature identifier symbol*?
32. Define *datums,* describe the *datum feature symbol,* and discuss how the datum feature symbol is shown on the drawing.
33. Identify the use of *datum targets* and explain how they are displayed on the drawing.
34. Briefly and in general name the tools used for creating custom symbols.
35. Define and describe a *revision history block* and a *revision symbol,* and give another name for each.

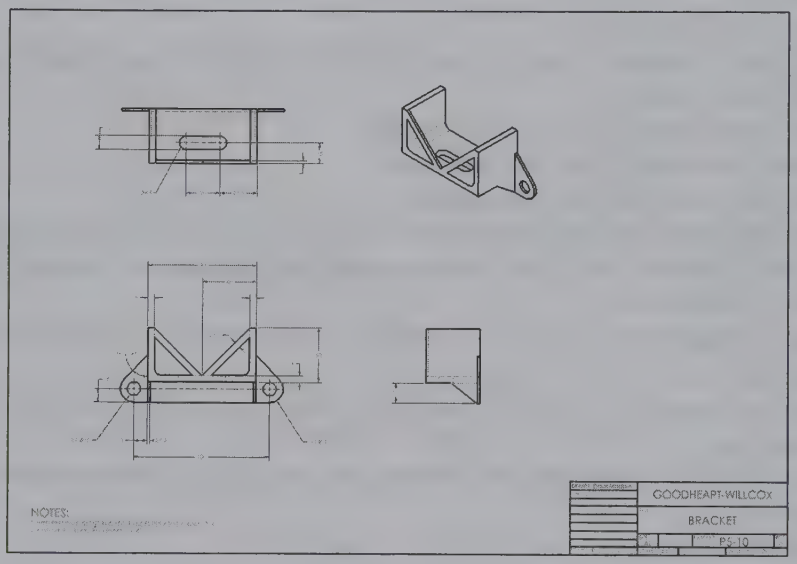
Problems

Instructions:

- Open the indicated drawing file.
- Save a copy of the file using the specified name.
- Close the original file, and open the copied file.
- Use the information provided to complete the drawing as shown.
- Drawing view scale is 1:1 unless otherwise noted.

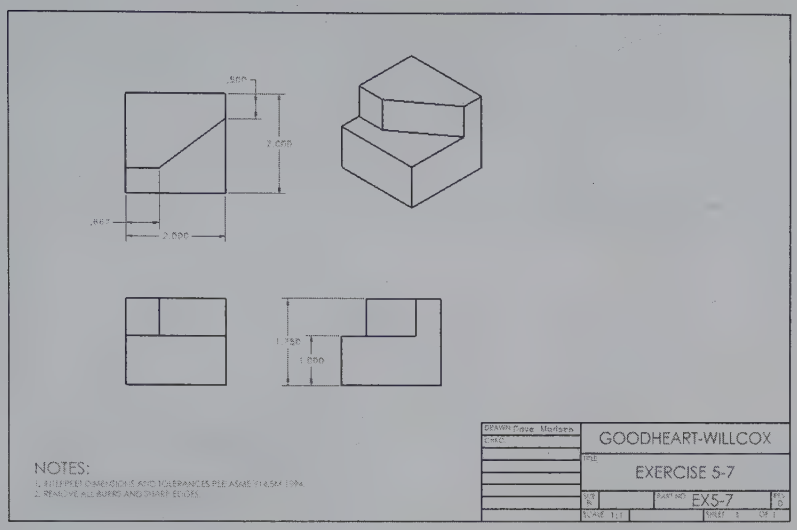
1. **Name:** Bracket
   Open File: EX17-1.idw
   Save Copy As: P18-1.idw
   Specific Instructions:
   - Add general notes and dimensions.

   The final drawing should look similar to the drawing shown.



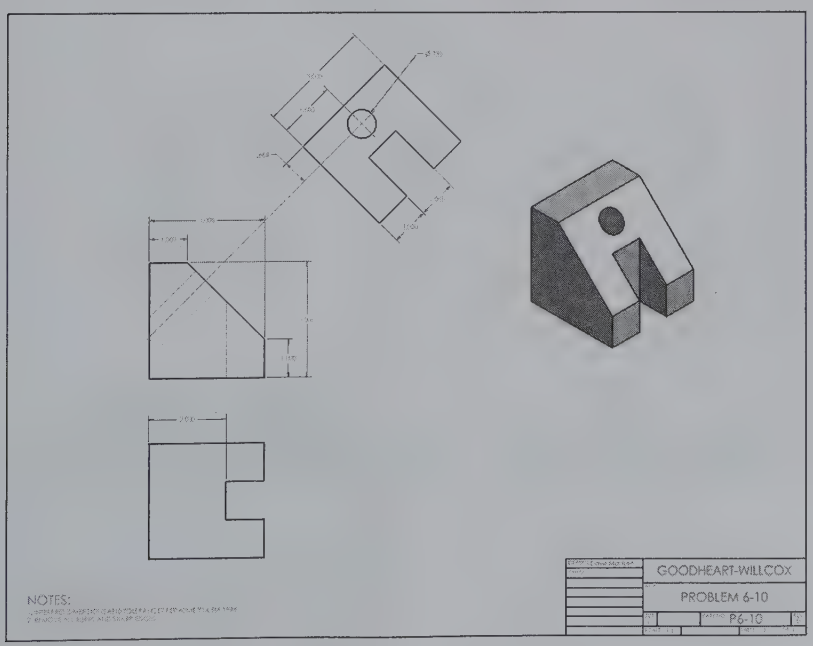
2. **Name:** Angle Block
Open File: EX18-5.idw
Save Copy As: P18-2.idw
Specific Instructions:
- Add general notes.

The final drawing should look similar to the drawing shown.



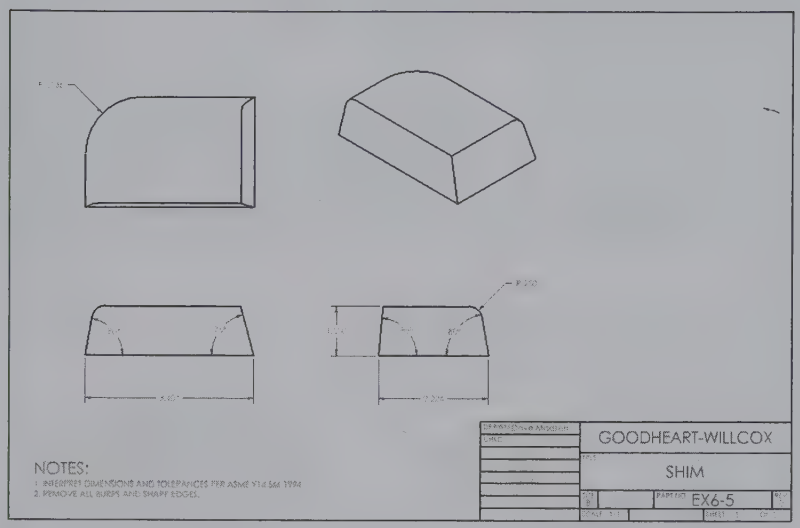
3. **Name:** Angle Bracket
Open File: EX18-6.idw
Save Copy As: P18-3.idw
Specific Instructions:
- Modify dimensions.
- Add a shaded isometric view.

The final drawing should look similar to the drawing shown.



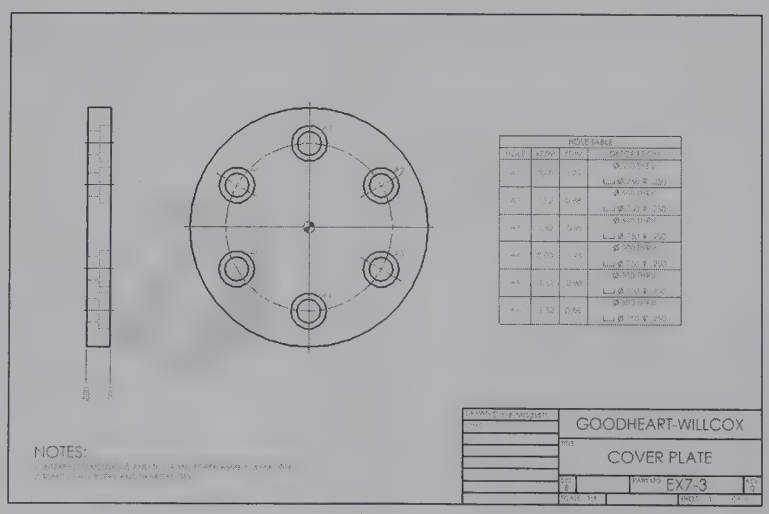
4. **Name:** Shim
Open File: EX17-10.idw
Save Copy As: P18-4.idw
Specific Instructions:
- Add general notes and dimensions.

The final drawing should look similar to the drawing shown.



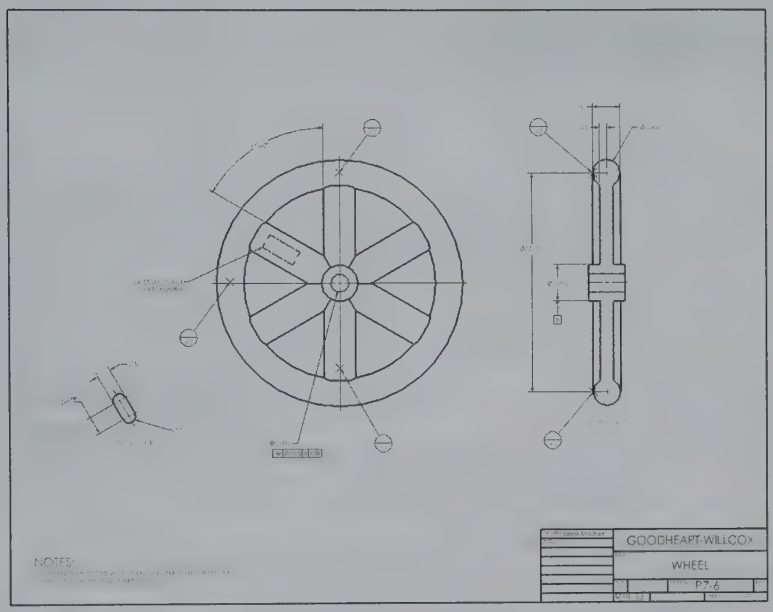
5. **Name:** Cover Plate
Open File: EX18-12.idw
Save Copy As: P18-5.idw
Specific Instructions:
- Place a border and title block.
- Add general notes and dimensions.

The final drawing should look similar to the drawing shown.



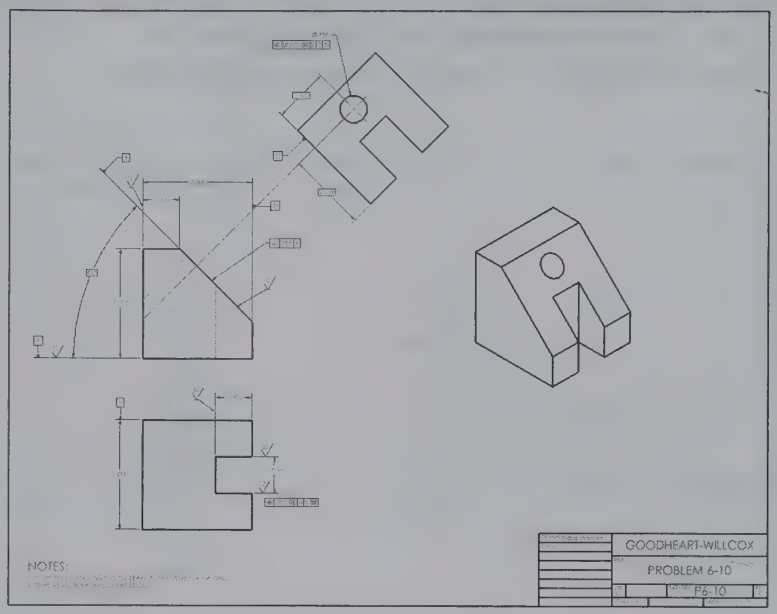
6. **Name:** Wheel
   Open File: EX17-5.idw
   Save As: P18-6.idw
   Specific Instructions:
   - Place a border and title block.
   - Change the scale of the base view to 1:2. The section view should automatically update to the correct scale.
   - Add SECTION B-B as shown.
   - Place general notes and dimensions. Note: Create the ink stamp as a draft view sketch, and then dimensioned with a leader note.

   The final drawing should look similar to the drawing shown.



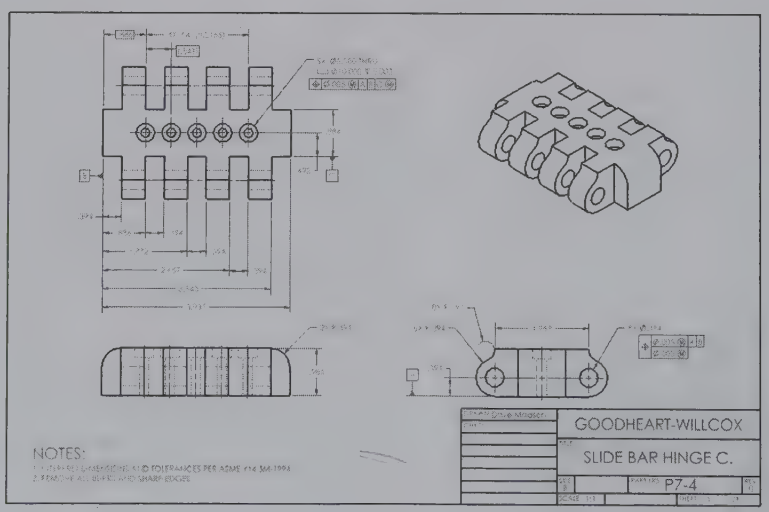
7. **Name:** Angle Bracket
Open File: EX18-6.idw
Save Copy As: P18-7.idw
Specific Instructions:
- Modify dimensions.
- Add dimensions.
- Add a hidden line removed isometric view.

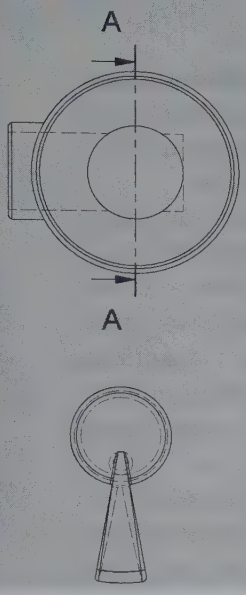
The final drawing should look similar to the drawing shown.

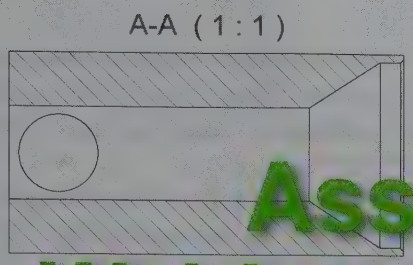


8. **Name:** Slide Bar Hinge Connector
Open File: P17-7.idw
Save As: P18-8.idw
Specific Instructions:
- Add general notes and dimensions.

The final drawing should look similar to the drawing shown.

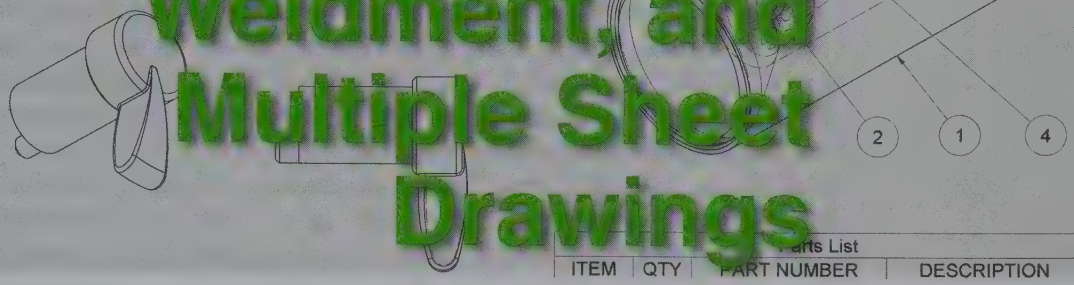






CHAPTER 19

Assembly, Weldment, and Multiple Sheet Drawings



Learning Objectives

After completing this chapter, you will be able to do the following:

- Create assembly drawing views.
- Add overlay drawing views.
- Create weldment drawing views.
- Add and modify balloons.
- Insert and modify parts lists.
- Use weldment model symbols and annotations.
- Place end fills and add caterpillars to weldment drawings.
- Place weld symbols.
- Create and work with multiple sheet drawings.
- Print drawings.

Assembly and weldment drawings can be generated quickly and easily from existing assembly models. Assembly drawings are produced in the same drawing work environment as part drawings. The same drawing tools used to create part drawings are used to place assembly and weldment views, and add projected, isometric, auxiliary, section, detail, conventional break, perspective, and broken out views. In addition, you can fully define the drawing with a variety of dimensions, notes, weld information, and text, including assembly balloons and parts lists.

This chapter explores the process of creating a complete set of ***working drawings***, which includes a fully defined assembly or weldment drawing with a parts list and a monodetail drawing for each part or multidetail drawing. A single drawing file, .idw, can be used to generate working drawings, because you can add multiple drawing sheets to a single file. For example, a single drawing file may contain a sheet with an assembly drawing and several additional sheets consisting of individual part drawings.

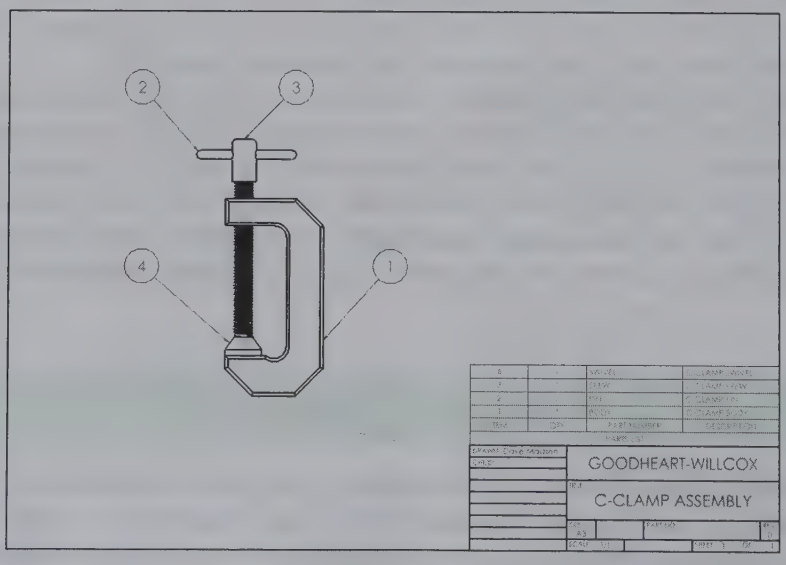
Assembly Drawings

An ***assembly drawing*** is a two-dimensional representation of an assembly. You can develop many different types of assembly drawings, depending on the application. Assembly drawings are created in a way that displays how each component relates to other components and the entire assembly. As a result, some assembly drawings may

only use a single front view of the assembly, while others may require multiviews, sections, and any other views needed to clearly define component and assembly characteristics. The following list represents a number of assembly drawings that you create using Inventor:

- A ***general assembly drawing*** is, as the name implies, the most common type of assembly drawing. This form of assembly drawing shows only the fully assembled product. A general assembly drawing may contain any views required to display each of the assembly components within the assembly, including multiviews, sections, and detail views. An example of a general assembly drawing is shown in **Figure 19-1.** General assembly drawings are created from existing assembly models and use balloons and a parts list to define components.
- A ***working drawing,*** or ***detail assembly drawing,*** is the same as a general assembly drawing in that they both show a product in a fully assembled form and may contain any views required to display each of the assembly components within the assembly. However, a working drawing, or detail assembly drawing, also displays details of each of the assembly components on the same page as the assembly drawing. These details may include dimensions, material information, and finishing specifications.
- An ***erection assembly drawing*** is the same as a general assembly drawing but also includes dimensions and fabrication specifications. Erection assembly drawings are created from existing assembly and part models. Dimensions, annotations, balloons, and a parts list are used to define component and assembly specifications.
- A ***subassembly drawing*** is typically a general assembly drawing, but could be a detail or erection assembly drawing. As you learned in previous chapters, subassemblies are groups of parts that are typically standard product items. Subassembly drawings do not show the entire assembled product, only the individual subassembly. In a set of working drawings, a subassembly drawing is followed by each of its own detail drawings. Subassembly drawings are

Figure 19-1.
An example of a two-dimensional assembly drawing created using Inventor.



created from existing assembly models and use balloons and a parts list to define components.

- A ***pictorial assembly drawing*** shows a pictorial view of an assembly, such as a single isometric view, instead of multiviews, as in a general assembly drawing. Usually a pictorial assembly drawing allows you to clearly show an assembled product for sales, maintenance, catalog, or other similar purposes. Pictorial assembly drawings are created from existing assembly models or presentations, and may or may not use balloons, a parts list, and other annotations to define components.
- An ***exploded assembly drawing*** is the same as a pictorial assembly drawing, but allows you to display assemblies in ways that are not shown when fully assembled. As with presentation models, assembly components are moved away from other components in an unassembled or ungrouped fashion. Exploded assembly drawings, which clearly show all assembly components, are typically used to help with assembly visualization, and for maintenance and assembly purposes. Exploded assembly drawings are created from existing presentations and may or may not use balloons, a parts list, and other annotations to define components.

As with part drawings, a typical assembly drawing sheet has a border, title block, and drawing views complete with dimensions and annotations. The process for setting up an assembly drawing sheet is the same as the process of developing a part drawing sheet. There are a number of methods for developing a set of working drawings using Inventor. For example, you may want to initially create all of the detail part drawings and then add an assembly drawing. Conversely, you may want to begin with an assembly drawing and then develop detail drawings. This chapter approaches the creation of a set of working drawings in the following order:

1. Open a template complete with predefined iProperties, drafting standards, document settings, sheet parameters, a border and a title block.
2. Adjust iProperties, drafting standards, document settings, border and title block characteristics, and sheet parameters according to individual drawing requirements.
3. Place and create the required assembly drawing views.
4. Add balloons and a parts list.
5. Add any other required annotations or drawing information.
6. Using a single drawing file, add additional detail drawings.

PROFESSIONAL TIP



You can add a parts list before balloons, or balloons before a parts list. Both techniques are effective.

NOTE



The procedure for developing an assembly drawing is very similar to creating a part drawing. The main difference is that you use an existing assembly or presentation file to generate the drawing, and then add balloons and a parts lists.

Creating Assembly Drawing Views

The typical method for creating assembly views is to use drawing view tools, though assembly views can be sketched or created by inserting views automatically into a sheet, using a sheet format. Drawing view tools allow you to insert a view from an existing file and create projected, isometric, auxiliary, section, detail, conventional break, perspective, and broken out views, just as when drawing parts. The only difference between using these tools to produce assembly views is that you select either an assembly or presentation file, depending on the application, when you access the **Base View** tool to place the initial assembly view, instead of selecting a part file, as when developing a part drawing.

Once you access the **Drawing View** dialog box, shown in Figure 19-2, using the **Base View** tool, the first step in creating an assembly drawing view is to select the existing assembly or presentation file from which you want to reference the drawing view. Pick the **Browse** button next to the **File** drop-down list to access the **Open** dialog box. Select either an existing assembly file (*.iam) or presentation file (*.ipn). An assembly file is used to create most assembly drawings, such as a general assembly drawing, while a presentation file is used to create a pictorial or exploded assembly drawing. Once you locate the file you want to use, pick the **Open** button to display the file and its directory in the **File** drop-down list.

The options provided in the **Drawing View** dialog box change according to the type of selected file. The **Representation** area is shown when placing an assembly drawing view. See Figure 19-2. The **Representation** area lists design view representations, positional representations, and level-of-detail settings created in the selected assembly model. This allows you to place an assembly drawing view based on a predefined or modified assembly model display. Pick the appropriate design view representation from the **View** list box, and pick the **Associative** check box if you want to associate the selected assembly design view representation with the drawing view. When you establish an associative relationship between the assembly design view representation and the drawing view, changes made to the assembly model design view representation are automatically displayed in the assembly drawing view. You can select a positional representation from the **Position** list box, and choose a preset level of detail from the **Level of Detail** list box. The preview of the selected assembly changes in the drawing window based on your selections.

When you select a presentation file, the **Presentation View** list box is shown. See Figure 19-3. The **Presentation View** drop-down list allows you to select a specific

Figure 19-2.
The **Drawing View** dialog box, **Component** tab as displayed when an assembly or weldment file is used to create the drawing view.

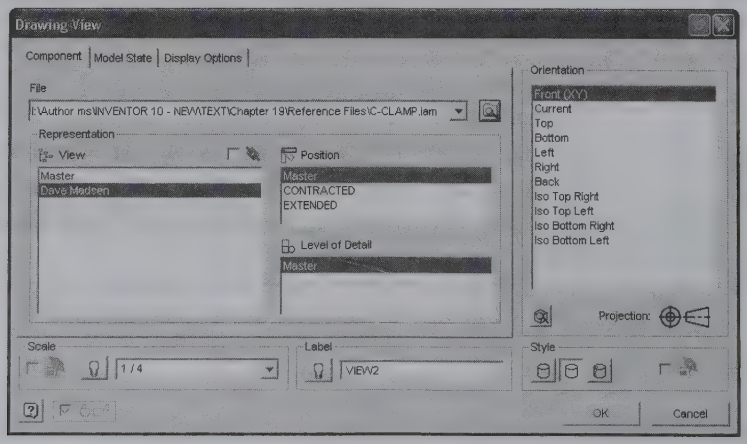
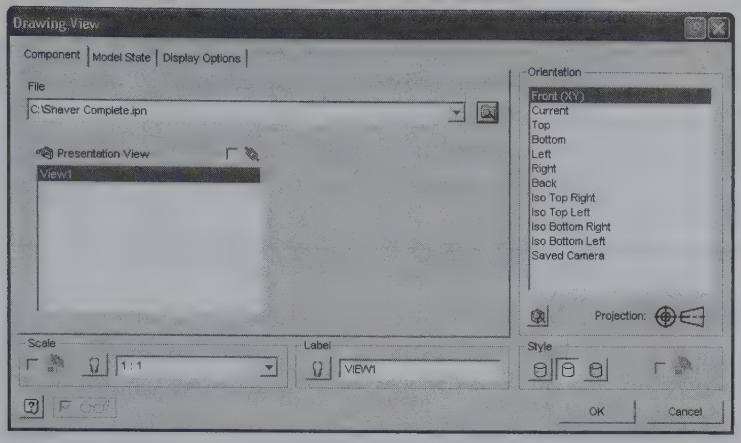


Figure 19-3.
The **Drawing View** dialog box, **Component** tab as displayed when a presentation file is used to create the drawing view.



presentation view to use for the assembly drawing base view. The **Associative** check box is also available when placing an assembly drawing view using a presentation file. Pick the **Associative** check box to associate the selected presentation view with the drawing view. When you establish an associative relationship between the presentation view and the drawing view, changes made to the presentation view in the model are automatically displayed in the drawing view.

Once you locate and select the desired assembly or presentation file, and choose the appropriate representation or presentation view options, the **Orientation**, **Scale**, **Label**, and **Style** areas function the same as when creating a part drawing. The **Model State** tab contains options specific to defining iPart, iAssembly, assembly, and weldment drawing base views. The **Weldment** area is described later in this chapter. The **Reference Data** area is used to define the display of assembly reference data when creating an assembly base. Reference data is an assembly component that is given a reference status in the assembly model bill of materials. Reference status components, and any child components associated with the reference item, are not displayed in the bill of materials. These components are included for reference purposes only. To create a reference component you must open the assembly model. Then, right-click on the component in the graphics window or the component name in the browser, and select the **Reference** option from the **BOM Structure** cascading submenu; or access the component iProperties, pick the **Occurrence** tab and select the **Reference** option from the **BOM Structure** drop-down list. Once a component is given reference status, you can choose how to show reference data in the drawing view using the following areas:

- **Line Style.** This area is used to define the display of reference data in the drawing view. Pick the **Reference Data** radio button to show reference data using phantom lines; choose the **As Part** radio button to display reference data normally, like other components; or select the **Off** radio button to hide reference data from the drawing view.
- **Hidden Line Calculation.** Pick the **Reference Data Separately** radio button to calculate reference data separately from other components in regard to the display of hidden objects. When selected, components hidden by reference data are shown using visible object lines. Choose the **All Bodies** radio button to calculate reference data the same as other components in regard to the display of hidden objects. When selected, components hidden by reference data are calculated and displayed as hidden objects, using hidden lines.

- **Margin.** This edit box is used to define the size of the boundary that determines how much of the reference data is shown in the drawing view. If reference data is not shown or is cut-off in the drawing, you must increase the margin depending on how much of the component you want to show.

NOTE



The **iPart Member** or **iAssembly Member** area of the **Model State** tab is a list box available when using iPart or iAssembly models to create a drawing view. Select the appropriate iPart or iAssembly member to use for a drawing.

The **Display Options** tab, shown and described in Chapter 17, contains check boxes and drop-down lists that correspond to items that can be displayed in the drawing view, depending on the type of file you selected in the **Component** tab. Once you specify all drawing view options, pick a location on the sheet where you want to place the base view.

NOTE



Once the initial assembly view has been placed using the **Base View** tool, you can use the **Projected View, Auxiliary View, Section View, Detail View, Broken View,** and **Break Out View** tools to create additional views.

PROFESSIONAL TIP



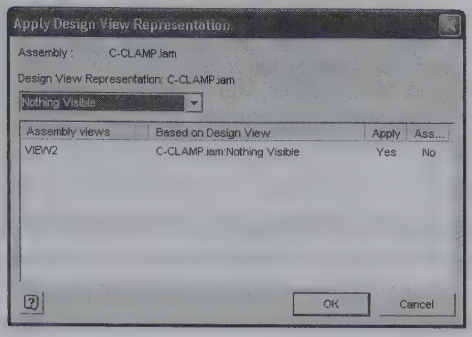
In order to create a single sectioned assembly view, place a base view such as a top view outside of the sheet. Then cut the base view and position the sectional view on the sheet.

Working with Assembly Drawing Views

Once you place the required assembly drawing views, some specific options that relate only to assembly drawings are available, in addition to the tools and techniques described throughout this book. These options allow you to change design view representations and work with assembly components. In addition, you may want to become familiar with the preference that allows you to exclude components from an assembly section view. The following information describes each of these functions:

- **Design view representations.** If you create an assembly drawing using a specified design view representation, you have the option of redefining the drawing with a different design view representation. To change the design view representation used for the drawing, right-click on an assembly drawing view in the graphics window, or an assembly drawing view in the browser. Then select the **Apply Design View...** menu option to open the **Apply Design View Representation** dialog box. See Figure 19-4. The **Apply Design View** dialog box displays the assembly and design view representation files currently used to generate the selected assembly drawing view. To specify a different design view representation, select a design view from the **Design View** drop-down list or pick the **Browse** button to locate a view. You will notice that all assembly-drawing views are listed in the **Assembly Views** list box, along with the design view representation used to create the drawing view. To apply the new design view representation selection to the current drawing view, the **Apply** option must be set to Yes. If you do not want to redefine the design view representation for a certain drawing view, pick

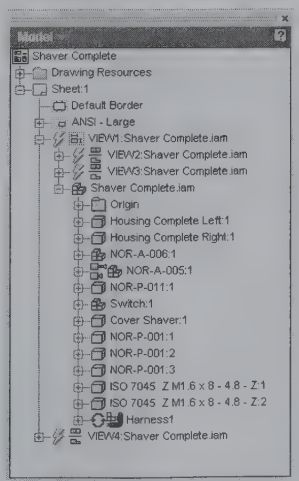
Figure 19-4.
The **Apply Design View Representation** dialog box.



the Yes option to change its value to No. Additionally, to associate the assembly model design view with the drawing view, as previously described, the **Associative** option must be set to Yes. Once you fully define the specifications in the **Apply Design View** dialog box, pick the **OK** button to change the design view representations used in the drawing.

- **Assembly components.** Typically, it is difficult to manipulate and access assembly drawing view preferences. For example, you cannot show assembly drawing model dimensions unless assembly components are available for selection. To access the components and features used to define the assembly and the assembly drawing, expand the assembly drawing view node in the browser, or right-click on the drawing view name in the browser and select the **Expand All Children** pop-up menu option. As you can see from Figure 19-5, the contents of an assembly drawing can be extensive depending on the number of assembly components and drawing views. Once the content of a drawing view is shown in the browser, you can work with and manipulate specific components for assembly drawing purposes.
- **Section views.** Assembly drawing section views are created the same way as part drawing section views, described earlier in this book. If you do not want to display the geometry of a certain component in a section display, such as the pin shown in Figure 19-6, you must first locate the component you do not want to show in the section view from the browser. Then, right-click on the component and select the **Section** menu option.

Figure 19-5.
Showing assembly drawing view contents.

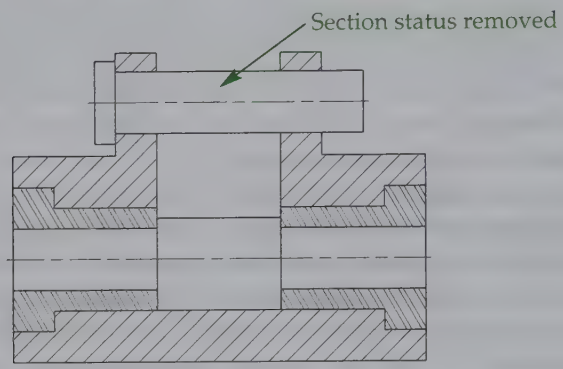


NOTE

You may also be able to control the sectioning characteristics of standard parts in an assembly section view by modifying the **Section Standard Parts** variable in the **Drawing View** dialog box, **Options** tab.

Exercise 19-1 Complete the Exercise on the Student CD.

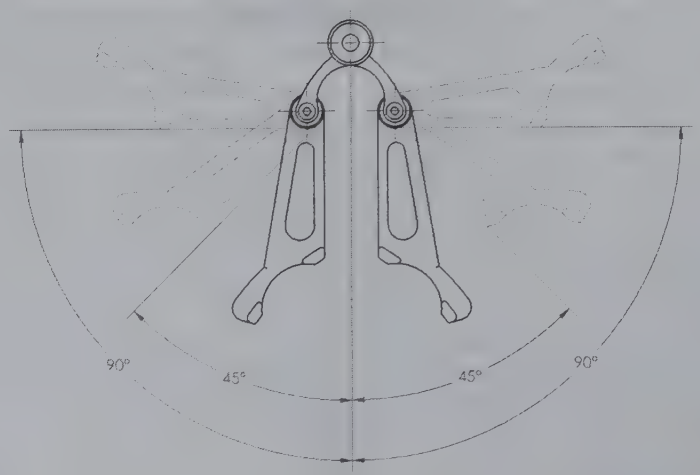
Figure 19-6.
Removing component section view priority.



Adding Overlay Drawing Views

Inventor ***overlay drawing views*** are used to create alternate position assembly drawing views. ***Alternate position drawing views***, also known as ***multiple position drawing views***, are used primarily to document the design of a product by showing an assembly in more that one position using a single sheet. Figure 19-7 shows an example of an Inventor overlay drawing view. As you can see, overlay drawing views are added to a standard assembly drawing view, such as a view created using the **Base View** tool. Overlay drawing views are typically shown using phantom lines, which clearly distinguishes alternate positions from the primary, or source, assembly position.

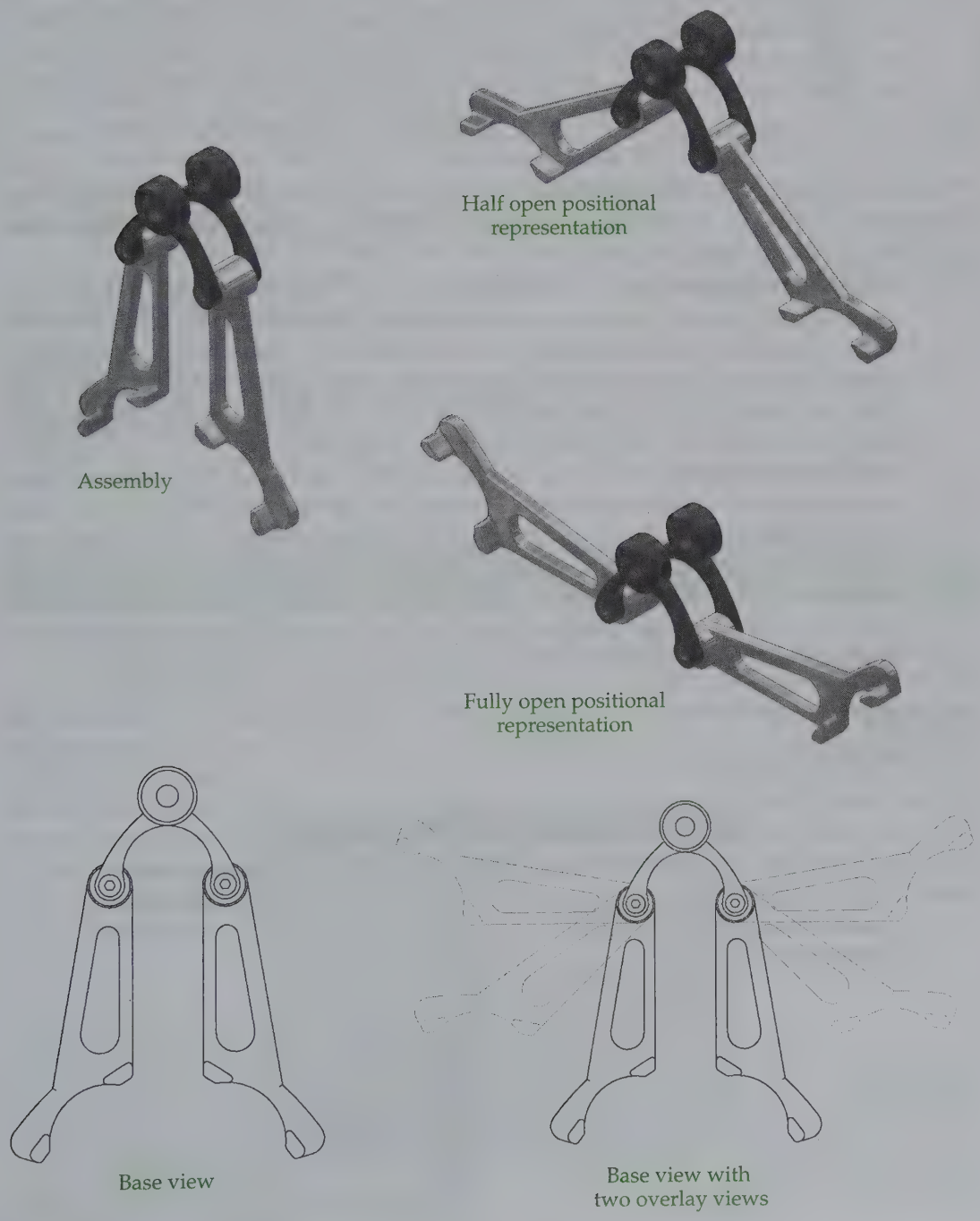
Figure 19-7.
An example of adding overlay drawing views to an assembly drawing.



An overlay drawing view is created by referencing a predefined assembly model positional representation and an existing assembly base, projected, or auxiliary view. As a result, in order to create an overlay drawing view you must have an assembly drawing view on the sheet that references an assembly model with at least one positional representation. Refer to previous chapters for information on assembly models and positional representations. The following steps outline the procedure for developing an overlay drawing view, which begins in the assembly model environment (see **Figure 19-8**):

1. Create an assembly model. The fully assembled and constrained model can be used as the initial drawing view to which overlay drawing views are added.
2. Add one or more positional representations that will later be used to generate overlay drawing views.

Figure 19-8.
The positional representations in an assembly are used to create overlay views.



3. Place an assembly base, projected, or auxiliary view.
4. Add one or more overlay drawing views.

Overlay drawing views are generated from existing base views or dependent projected or auxiliary views that contain positional representations. The **Overlay View** tool allows you to add as many overlay views as needed, depending on the number of positional representations available in the parent assembly model. You can access the **Overlay View** tool anytime during drawing development by picking the **Overlay...** button on the **Drawing Views Panel** bar; picking the **Overlay View** button on the **Drawing Views Panel** toolbar; right-clicking an existing parametric drawing view in the graphics window, and selecting **Overlay...** from the **Create View** cascading submenu; right-clicking an existing parametric drawing view name in the browser, and select **Overlay...** from the **Create View** cascading submenu; or from the **Insert** pull-down menu, select the **Overlay...** option of the **Model Views** cascading submenu.

NOTE

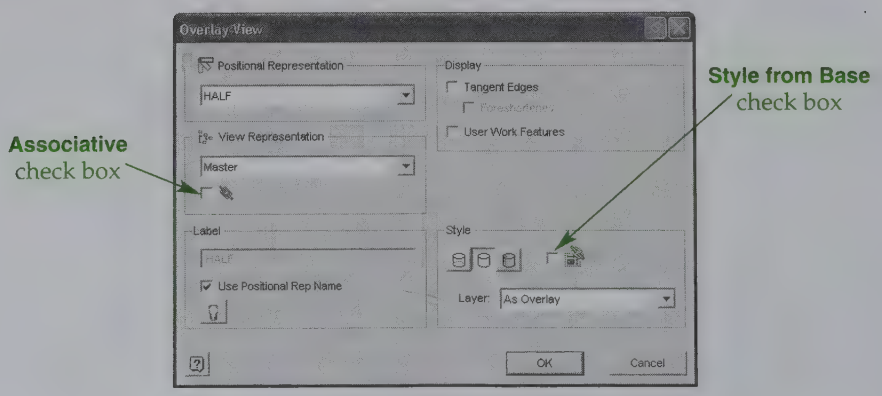


If you access the **Overlay View** tool from the **Drawing Views Panel** bar, **Drawing Views Panel** toolbar, or the **Insert** pull-down menu, you must select the view from which you want to create an overlay.

When you access the **Overlay View** tool and a view is selected, the **Overlay View** dialog box is displayed. See **Figure 19-9.** Use the following options in the **Overlay View** dialog box to define the overlay view:

- **Positional Representation.** This area contains a drop-down list that displays the assembly model positional representations available to use for the overlay view. Again, an overlay drawing view is created by referencing a predefined assembly model positional representation. Only positional representations that are found in the parent assembly model, and that have not already been used in the drawing view, are available for selection.
- **View Representation.** This area includes a drop-down list that allows you to select a specific design view representation to use for the overlay view. Then, pick the **Associative** check box if you want to associate the selected assembly design view representation with the overlay view. When you establish an associative relationship between the assembly design view representation and the overlay view, changes made to the assembly model design view representation are automatically displayed in the assembly drawing.

Figure 19-9.
The **Overlay View** dialog box.



- **Label.** This area allows you to modify the overlay view label. You can use the existing positional representation name by selecting the **Use Positional Rep Name** check box, or define a different name by deselecting the **Use Positional Rep Name** check box, and entering a new name in the **Label** edit box. Pick the **Toggle Label Visibility** button if you want the label displayed in the view.
- **Display.** This area contains three check boxes that correspond to items that can be displayed in the overlay view. Pick the **Tangent Edges** check box to display tangent model edges. You can also select the **Foreshortened** check box to shorten tangent edges in an effort to differentiate tangent edges from visible object lines, assuming you do not use a separate tangent edge layer. Select the **User Work Features** check box if you want to show work features placed in the selected model. Only work features that are normal to the specified view orientation are displayed.
- **Style.** This area allows you to specify the overlay view display. You can choose to display the view with hidden lines by picking the **Hidden Lines** button, without hidden lines by picking the **Hidden Lines Removed** button, or as shaded by picking the **Shaded** button. Or, pick the **Style from Base** check box to set the overlay view style according to the selected assembly view style. A **Layer** drop-down list is also available to select the layer assigned to overlay view geometry. For most applications, the **As Overlay** layer option should be selected, which uses the layer specified in **Overlay View** object type of the **Object Defaults Styles** area of the **Styles and Standards Editor**. However, you can pick the **As Parts** layer option to use the same layers applied to drawing views.

Once you specify the options in the **Overlay View** dialog box, pick the **OK** button to create the overlay drawing view.



Complete the Exercise on the Student CD.

Weldment Drawings

A *weldment drawing* is a 2D representation of a weldment. An Inventor weldment model is used to show how multiple part and subassemblies are welded together. The model can then be used to produce a 2D weldment drawing, which also shows how components are welded together. A weldment is a welded assembly, which means a weldment drawing is essentially a type of assembly drawing. All the assembly drawing tools and development techniques previously described apply to working with weldment drawings. The major difference between an assembly drawing and a weldment is when you place a weldment drawing base view you can identify which design stage (assembly, preparation, weld, or machining) you want to display, in addition to a specific design view. In addition, you can automatically display weldment model symbols and annotations, and can add weldment drawing annotations including weld symbols, caterpillars, and end fills.

Creating Weldment Drawing Views

Weldment drawing views are created in the same way as part and assembly drawing views using sketch tools, sheet formats, and drawing management tools. However, when you insert the initial weldment drawing view using the **Base View** tool and the **Drawing View** dialog box, instead of picking a part, assembly, or presentation file, you select an existing weldment application of an assembly file (weldment model). The

Representation area is shown when placing a weldment drawing view. This area is the same as when placing a standard assembly, except the **Position** list box is unavailable because weldment components are fixed.

Once you locate and select the desired weldment file, and choose the appropriate representation view options, the **Orientation**, **Scale**, **Label**, and **Style** areas function the same as when creating a part or assembly drawing. The **Model State** tab contains a **Weldment** area, in addition to the **Reference Data** area previously described. See Figure 19-10.

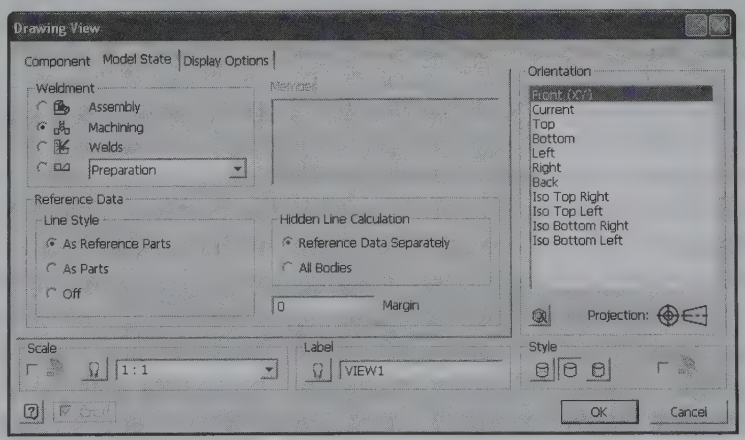
The **Weldment** area is used to select a weldment development stage, including assembly, preparation, welds, and machining. The design stage you choose from the **Weldment** area is represented in the base drawing view. The following information briefly describes each option available from the **Weldment** area:

- **Assembly.** Select this option to create an assembly drawing view with no preparations, welds or machined features.
- **Machining.** Select this option to create a drawing of the fully assembled, prepared, welded, and machined weldment.
- **Welds.** Pick this option to create an assembly drawing view with preparations and welds, but no machined features.
- **Preparations.** Select this option to produce an assembly drawing view with prepared components but no weld or machined features. When you choose the **Preparations** option, the **Components** drop-down list becomes available. You can use this drop-down list to create a view of the entire prepared assembly, by pick in the **Assembly** option, or you can choose individual components to create a drawing view of a specific prepared assembly component.

The **Display Options** tab, shown and described in Chapter 17, contains check boxes and drop-down lists that correspond to items that can be displayed in the drawing view, depending on the type of file you selected in the **Component** tab.

Once you specify all drawing view options, pick a location on the sheet where you want to place the base view.

Figure 19-10.
The **Drawing View** dialog box, **Model State** tab.



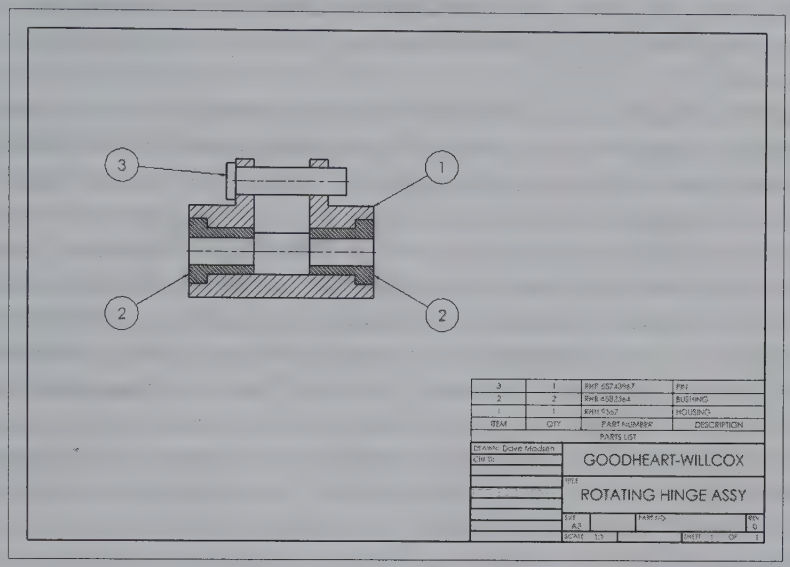
Dimensioning Assembly and Weldment Drawings

The amount and type of dimensions and annotations added to an assembly or weldment drawing depends on the application. For example, an erection assembly drawing contains dimensions and fabrication specifications, while a pictorial assembly drawing

may have no annotations. Still, most assembly drawings do have balloons and a parts list, and a weldment drawing may include end fills, caterpillars, and weld symbols.

In general, a ***parts list*** records and displays the parts and subassemblies used to create the assembly. Then, ***balloons*** link the parts list to the drawing, by identifying the assembly components in relation to the parts list specifications. See **Figure 19-11.** You can add a parts list before you place balloons, or create balloons before you add a parts list. Both techniques are effective, but this chapter introduces balloons first, followed by parts lists.

Figure 19-11.
An example of an assembly drawing with a parts list and balloons.



Adding Balloons

The term ***balloon*** refers to a shape, usually a circle, connected to an assembly component with a leader. Inside the balloon is an identification number, typically known as the item number. The name *item number* implies that numbers are used to define assembly components. However, some companies choose to use letters. Balloons allow you to reference assembly components to the parts list. As a result, when you change the order and value of item numbers in a parts list, the corresponding balloon identification number changes to reflect the parts list modification. In addition, because balloons are connected to a parts list, you can either place balloons before you add the parts list, or place balloons after you insert the parts list.

NOTE



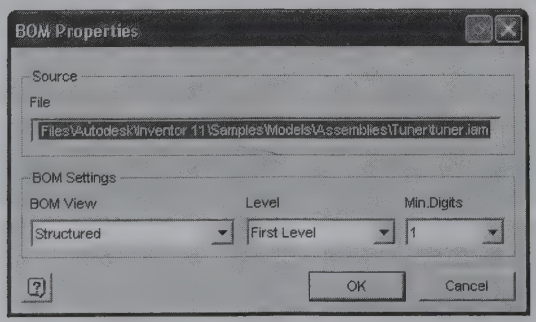
Individual balloon characteristics can be modified once they are placed, but all balloons and balloon leaders in the drawing should have the same characteristics.

There are two different options for adding balloons to an assembly drawing. You can place balloons individually using the **Balloon** tool, or automatically add balloons to the entire drawing using the **Auto Balloon** tool. To add assembly drawing balloons individually, access the **Balloon** tool by picking the **Balloon** flyout tool button from the **Drawing Annotation Panel** bar; picking the **Balloon** flyout tool button from the **Drawing Annotation Panel** toolbar; pressing the [B] key on your keyboard; or picking the **Balloon** button on the **Drawing Balloon** toolbar.

After you access the **Balloon** tool, use your cursor to select a point where you want the balloon leader to begin. If a parts list has not already been inserted, the **BOM Properties** dialog box pops up. See Figure 19-12. The **BOM Properties** dialog box opens when you try to create a balloon in a drawing that does not have a parts list, because balloons and parts lists reference each other. Inventor calculates assembly model components using a bill of materials (BOM). The bill of materials is later referenced in the drawing environment to create balloons and parts lists. The **Source** area of the **BOM Properties** dialog box contains the **File** display box, which identifies the location of the assembly or presentation file associated with the selected drawing view, and corresponding bill of materials. The **BOM Settings** area is used to define which components are referenced from the assembly model bill of materials, and how the components are displayed. The options in the **BOM Settings** area change according to the bill of materials view selected from the **BOM View** drop-down list. The following describes each view:

- **Structured.** Pick this option to set the **Structured** bill of materials view and display the **Level** drop-down list. Choose **First Level** from the **Level** drop-down list to reference and display all first level parts and subassemblies. This includes the assembly parts and subassemblies, but not the parts and subassemblies used to build the subassemblies. When **First Level** is selected, the **Min. Digits** drop-down list is provided allowing you to select a value from 1 to 6 according to how many digits you want to use for item numbering. For example, item 2 will display as 2 if you select 1 as the minimum digit, while selecting 3 as the minimum digit will display item 2 as 002. Choose **All Levels** from the **Level** drop-down list to reference and display all the parts and subassemblies found in the assembly file, with the subassembly parts nested under the subassembly. Subassemblies contain multiple parts or even smaller subassemblies. As a result, when **First Level** is selected the **Delimiter** drop-down list is provided, allowing you to select or enter a delimiter that is used to identify subassembly components. For example, subassembly parts are listed as 1.1, 1.2, 1.3, and so on, in the parts list. You can change the subassembly delimiter, which is a period (.) using the previous example, by entering or selecting a new character, such as a hyphen (-) or asterisk (*) from the **Delimiter** drop-down list.
- **Parts Only.** Pick this option to set the **Parts Only** bill of materials view and display the **Numbering** drop-down list. A parts-only view lists only the parts found in the assembly file, including the parts used to create subassemblies, but does not actually display subassemblies. Choose **Numeric** from the **Numbering** drop-down list to use numbers when identifying parts. When **Numeric** is selected, the **Min. Digits** drop-down list is provided allowing you to select a value from 1 to 6 according to how many digits you want to use for item numbering. Choose **Alpha** from the **Numbering** drop-down list to use letters when identifying parts.

Figure 19-12.
The **BOM Properties** dialog box.



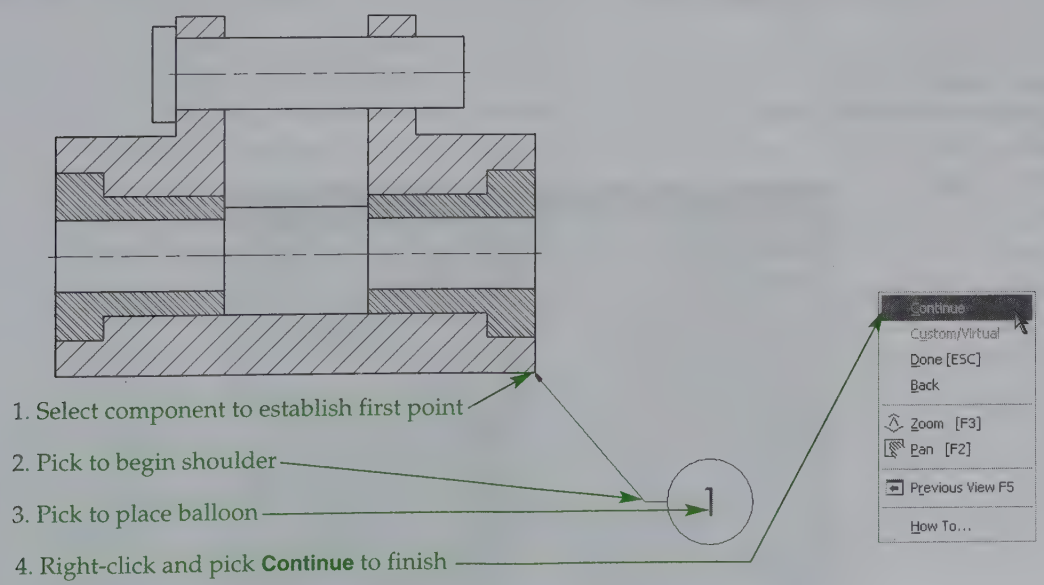
When **Alpha** is selected the **Case** drop-down list is provided, allowing you to select **Upper** or **Lower** depending on whether you want to use uppercase or lowercase letters.

- **Structured (legacy).** Pick this option to use the structured bill of materials format, available with previous versions of Inventor. The **Structured (legacy)** bill of materials view references and displays all the parts and subassemblies found in the assembly file, with the subassembly parts nested under the subassembly. Pick the **Inheritance** check box if you want the display characteristics of subassembly components to match the display characteristics of the parent subassembly in the parts list. For example, if you change an assembly item number from 1 to A, the subassembly components change from 1.1, 1.2, and 1.3 to A.1, A.2, and A.3. The **Delimiter** drop-down list is also provided, allowing you to select or enter a delimiter that is used to identify subassembly components.
- **Parts Only (legacy).** Pick this option to use the parts-only bill of materials format available with previous versions of Inventor. The **Parts Only (legacy)** bill of materials view references and displays only the parts found in the assembly file including the parts used to create subassemblies, but does not actually display subassemblies. The **Inheritance** check box and **Delimiter** drop-down list are also provided.

Once you define the options in the **BOM Properties** dialog box, pick the **OK** button to continue the process of adding balloons to the assembly drawing.

The point on the assembly you select to connect the leader and balloon defines both the selected component and the leader start point. The next step in defining the balloon is to drag the balloon to the desired location and pick the position to specify the leader length. Depending on the leader style, you can continue selecting points as required, such as adding a leader shoulder. See **Figure 19-13**. After you select a point on the appropriate drawing geometry and locate the balloon, press [Enter], or right-click and select the **Continue** menu option to create the balloon. The **Balloon** tool is still active, allowing you to continue placing balloons as required. To exit the **Balloon** tool, right-click and select the **Done** menu option, press the [Esc] key on your keyboard, or access a different tool.

Figure 19-13.
Placing a balloon using the **Balloon** tool.



If you would rather add all the required assembly drawing balloons at one time, using a single selection, access the **Auto Balloon** tool. Do this by picking the **Auto Balloon** flyout tool button from the **Drawing Annotation Panel** bar; picking the **Auto Balloon** flyout tool button from the **Drawing Annotation Panel** toolbar; or picking the **Auto Balloon** button on the **Drawing Balloon** toolbar.

The **Auto Balloon** dialog box, shown in **Figure 19-14**, is displayed when you access the **Auto Balloon** tool. To add balloons, first pick the **Select View Set** button if it is not currently active, and then choose the drawing view to which you want to add balloons. Once a view is selected, the **Add or Remove Components** button should become active, allowing you to select each assembly or weldment component that will receive a balloon. You can pick components in the graphics window or the browser, and deselect components by holding down the [Ctrl] key on you keyboard and choosing the selected component. By default, the **Ignore Multiple Instances** check box is selected, which does not allow multiple instances of components to be individually ballooned. For example, if an assembly contains three of the same bolts, only one bolt will be ballooned if the **Ignore Multiple Instances** check box is selected. Deselect the **Ignore Multiple Instances** check box to add balloons to all assembly components.

The next step when adding balloons using the **Auto Balloon** dialog box is to define how the balloons are arranged around the assembly view. Pick the **Around** radio button in the **Placement** area to place the balloons around the assembly in a rectangular configuration, select the **Horizontal** radio button to place all the balloons horizontally aligned with each other the distance specified in the **Offset Spacing** edit box, or choose the **Vertical** radio button to place all the balloons vertically aligned with each other the distance specified in the **Offset Spacing** edit box. **Figure 19-15** shows examples of each placement option. Then, pick the **Select Placement** button and choose the location on the sheet where you want to place the balloons.

Once you select an assembly view, pick assembly components, and define the balloon placement, you are ready to generate the balloons. However, you can specify additional balloon options using the **BOM Settings** and **Style Overrides** areas. The **BOM Settings** area includes the same settings available in the **BOM Properties** dialog box. The **Style Overrides** area is used to override the default balloon display characteristics. In order to override the default balloon settings, you must select the **Balloon Shape** check box. Then, the balloon type buttons and the **Symbols** drop-down list become available. The **Style Overrides** area balloon type buttons are the same as those listed in the **Balloon Style** area of the **Style and Standards Editor**. Select the desired balloon type or pick the **User-Defined Symbol** button and choose a sketched symbol from the

Figure 19-14.
The **Auto Balloon** dialog box.

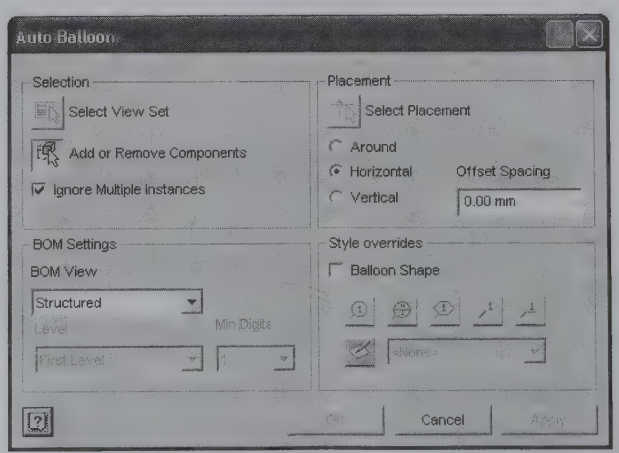
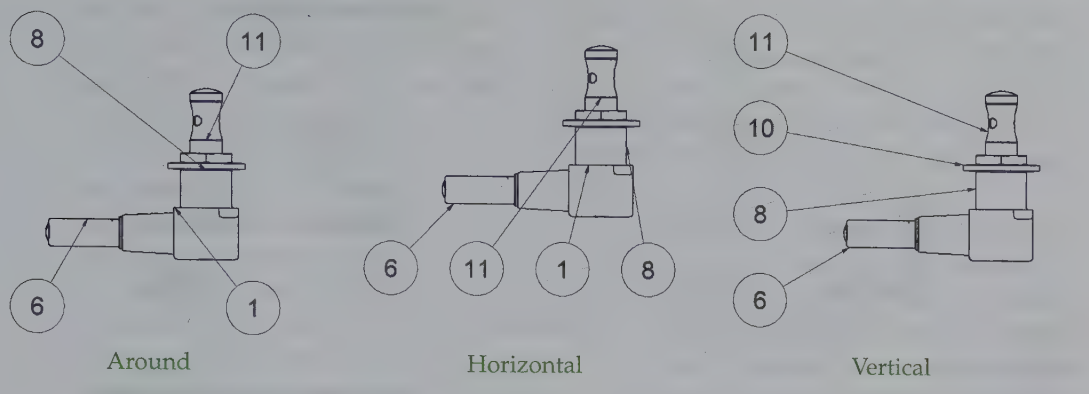


Figure 19-15.
These are the three options available when placing balloons with **Auto Balloon**.



Symbols drop-down list. The symbols available from the **Symbols** drop-down list correspond to drawing sketched symbols.

Once you select a drawing view, assembly components, balloon placement options, and balloon display characteristics, pick the **Apply** button to create the balloons, and continue using the **Auto Balloon** tool, or select the **OK** button to create the balloons and exit the **Auto Balloon** tool.

Modifying Balloons

Typically, when you use the **Balloon All** tool, balloons are randomly placed on the drawing. In addition, even if you use the **Balloon** tool to individually place balloons, you may have to reorganize the balloons so they are arranged in an easy-to-read pattern. To redefine balloon location, select the balloon you want to modify. Then drag the green dots to the desired location. **Figure 19-16** shows how to reposition balloons so that they are correctly aligned and easy to read.

In addition to repositioning balloons and balloon leaders, use the following information to add or further manipulate balloon specifications:

- **Remove a balloon.** To remove an existing balloon from the drawing, right-click on a balloon and select the **Delete** menu option. Or, pick the balloon and press the [Delete] key on your keyboard.
- **Edit individual balloon arrowheads.** Typically, all balloon leader arrowheads in a drawing should be the same. However, if you want to change the leader arrowhead style of a specific balloon leader, right-click on the leader you want to modify, and select the **Edit Arrowhead...** pop-up menu option to display the **Change Arrowhead** dialog box. See **Figure 19-17**. To specify a different arrowhead style, select the arrowhead from the drop-down list, and pick the **Apply** button.
- **Group balloons.** Balloons may be grouped together for closely related clusters of assembly components, such as a bolt, a washer, and a nut. As shown in **Figure 19-18**, ***grouped balloons*** share the same leader, which is typically connected to the most obviously displayed component, such as the bolt mentioned in the previous example. To group two or more balloons, right-click on an existing balloon to which you want to add balloons, and pick the **Attach Balloon** pop-up menu option. Then, select an additional component in the drawing view or from the browser. The new balloon is attached to the existing balloon, but the final step is to pick the side where you want to place the new balloon in reference to the existing balloon. If you want to remove an attached balloon, right-click on the balloon you want to delete and select the **Remove Balloon** menu option, which is only available if two or more balloons are attached to each other.

Figure 19-16.
The position of balloons may not be optimal after using the **Auto Balloon** tool.

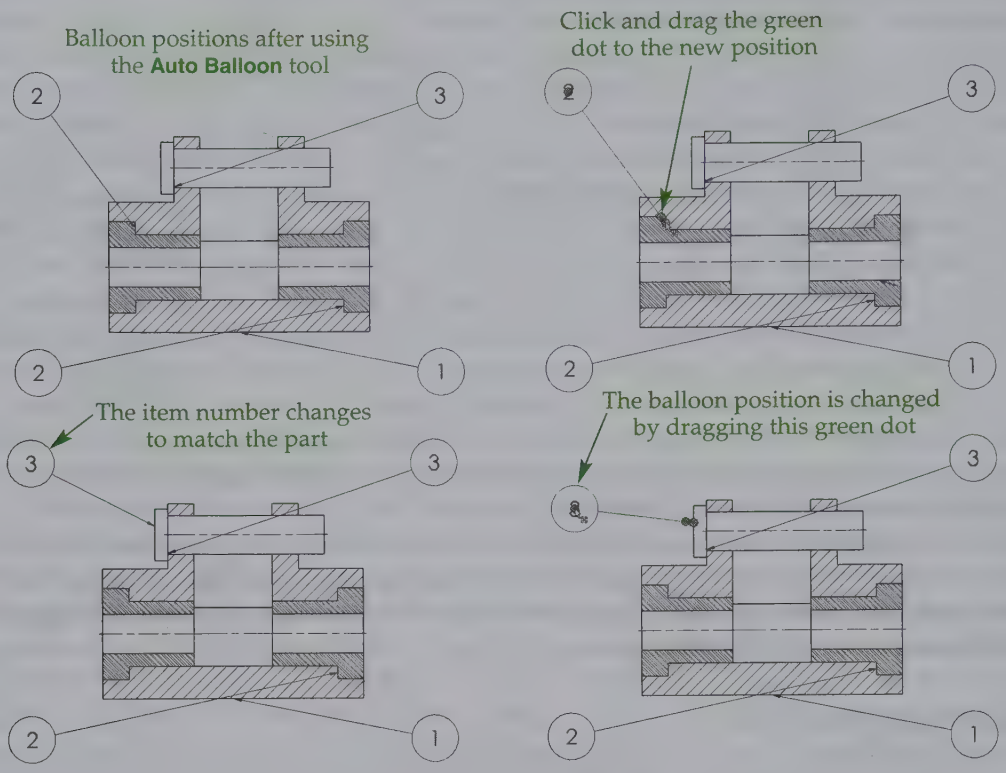
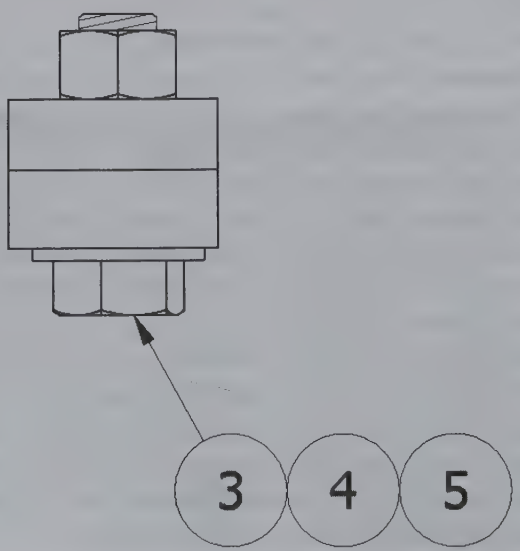


Figure 19-17.
The **Change Arrowhead** dialog box.



Figure 19-18.
These grouped balloons identify the bolt, nut, and washer in this view.



- **Place an additional segment or vertex.** Although usually only one leader is attached to a balloon, or group of balloons, you can add multiple leaders. To place an additional leader, right-click on the existing balloon or balloon leader and select the **Add Vertex/Leader** pop-up menu option. Then, pick the start point of the new leader, followed by selecting a point on the existing balloon to which you want to connect the new leader. See **Figure 19-19.** If you want to remove a leader, right-click on the balloon, and select the **Delete Leader** menu option, which is only available if two or more leaders are attached to a balloon, and then pick the leader you want to remove.
- **Edit individual balloons.** In some situations you may want to adjust specific balloons without redefining default settings. To edit a single balloon, right-click on the balloon you want to modify and select the **Edit Balloon...** pop-up menu option, to display the **Edit Balloon** dialog box. See **Figure 19-20.** In order to override the default balloon settings, you must deselect the **Override Shape (by style)** check box. Then, the balloon type buttons and the **Symbols** drop-down list become available. The **Edit Balloon** dialog box balloon type buttons are the same as those listed in the **Balloon Style** area of the **Style and Standards Editor**. Select the desired balloon type or pick the **User-Defined Symbol** button and choose a sketched symbol from the **Symbols** drop-down list. The symbols available from the **Symbols** drop-down list correspond to drawing sketched symbols. The **Edit Balloon** dialog box is also used to change balloon contents using the options in the **Balloon Value** area. You can override the current balloon value and the corresponding value in the parts list by selecting and entering a new value in the **Item** edit box. Another option is to change just the balloon value by selecting and entering a value in the **Override** edit box. When you finish redefining the selected balloon, pick the **OK** button to generate the changes and exit the **Edit Balloon** dialog box.
- **Align views.** Typically balloons are aligned in an orderly fashion, as shown in **Figure 19-21.** Though you can drag balloons to the desired location, there are specific options that allow you to perfectly align two or more balloons. To align balloons, hold down the [Ctrl] key on your keyboard, and select the balloons you want to align. Then right-click and select one of the following options from the **Align** pop-up cascading submenu:

Figure 19-19.
Adding a leader to a balloon.

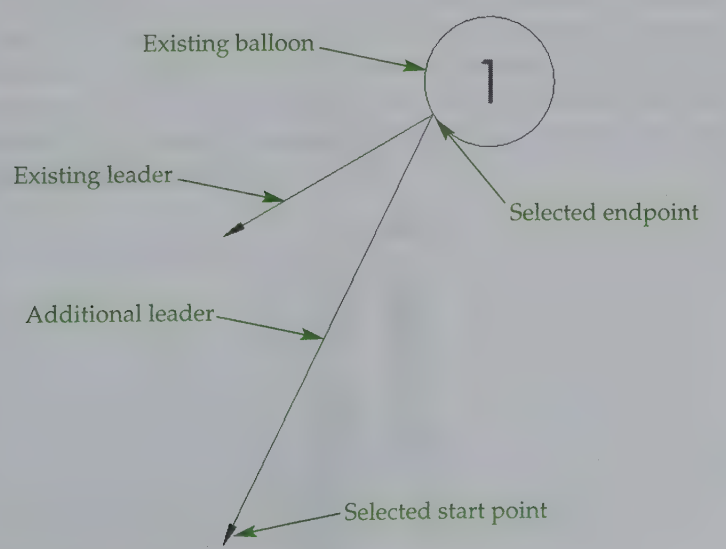
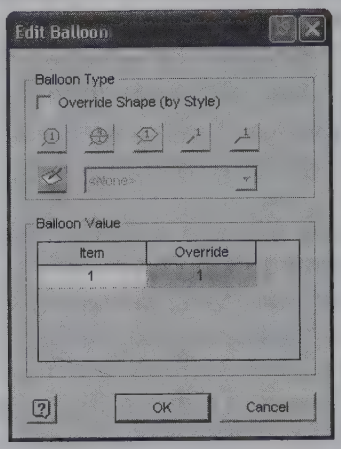
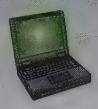


Figure 19-20.
The **Edit Balloon** dialog box.



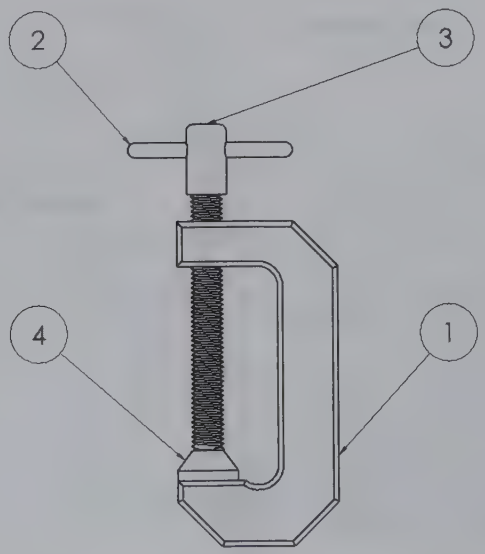
- **Vertical.** Pick this option to align the selected balloons vertically.
- **Horizontal.** Select this option to align the selected balloons horizontally.
- **Vertical Offset.** Choose this option to align the selected balloons vertically, using the offset distance specified in the **Balloon Style** area of the **Style and Standards Editor**.
- **Horizontal Offset.** Pick this option to align the selected balloons horizontally, using the offset distance specified in the **Balloon Style** area of the **Style and Standards Editor**.

PROFESSIONAL TIP



Autoproject lines used when automatically constraining sketch geometry can also be used to align balloons.

Figure 19-21.
Balloon alignment options.



Exercise 19-3 Complete the Exercise on the Student CD.

Exercise 19-4 Complete the Exercise on the Student CD.

Inserting Parts List

A *parts list* records and displays the parts and subassemblies used to create an assembly, and keys or links balloons to component information. See **Figure 19-22.** Parts list content differs greatly depending on the application and specific company standards. A typical parts list may contain some of or all the following information in a table format, with columns and rows:

- **Item Number.** This column contains item numbers, linked to the balloons, for each assembly components, and is typically the first parts list column.
- **Quantity.** This column identifies the number of components, or items, used in the assembly.
- **Part Number.** Use this column to display component part numbers. Part numbers are referenced to individual detail drawings, and may be used in addition to, or substituted by, the sheet or drawing number.
- **Drawing Number.** This column displays the drawing number on which the corresponding detail drawing is found. The drawing number may be used in addition to, or substituted with, the part number previously described.
- **Description.** Use this column to provide a description of each item.
- **Material.** This column identifies the material used to manufacture the part.
- **Additional columns.** A parts list may also contain additional company-specific columns, such as vendor and part purchasing information.

Inventor parts lists are similar to drawing title blocks, because a great deal of parts list information is automatically entered in the columns based on assembly and bill of materials characteristics. For example, if three of the same items are displayed in an assembly, a quantity of 3 is automatically established in the quantity area for the item in the parts list. Another example is how part numbers are referenced from the **Part Number** edit box of the **Properties** dialog box for each component file. Other parts list entries are also controlled by specific component properties set in the **Properties** dialog box. For example, if you enter Housing in the **Description** edit box of the **Properties** dialog box, Housing is displayed in the parts list description area for the item.

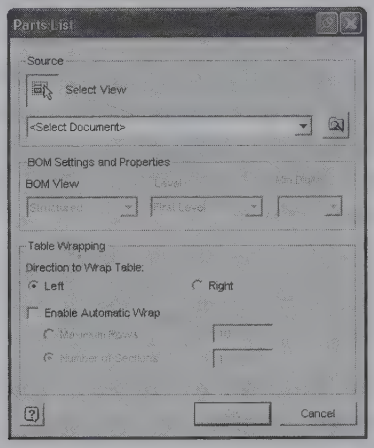
To insert an assembly drawing parts list, access the **Parts List** tool by picking the **Parts List** button on the **Drawing Annotation Panel** bar, or picking the **Parts List** button on the **Drawing Annotation Panel** toolbar.

Figure 19-22.
An example of a parts list.

PARTS LIST			
ITEM	QTY	PART NUMBER	DESCRIPTION
1	1	RHH 9567	HOUSING
2	2	RHB 4582364	BUSHING

The **Parts List** dialog box, shown in **Figure 19-23**, is displayed when you access the **Parts List** tool. To add a parts list, first pick the **Select View** button if it is not currently active, and then choose the drawing view from which you want to reference parts list information. An alternative option is to choose a file from which to reference parts list data by picking a file from the **Select Document** drop-down list or using the **Browse** button and the **Open** dialog box.

Figure 19-23.
The **Parts List** dialog box.



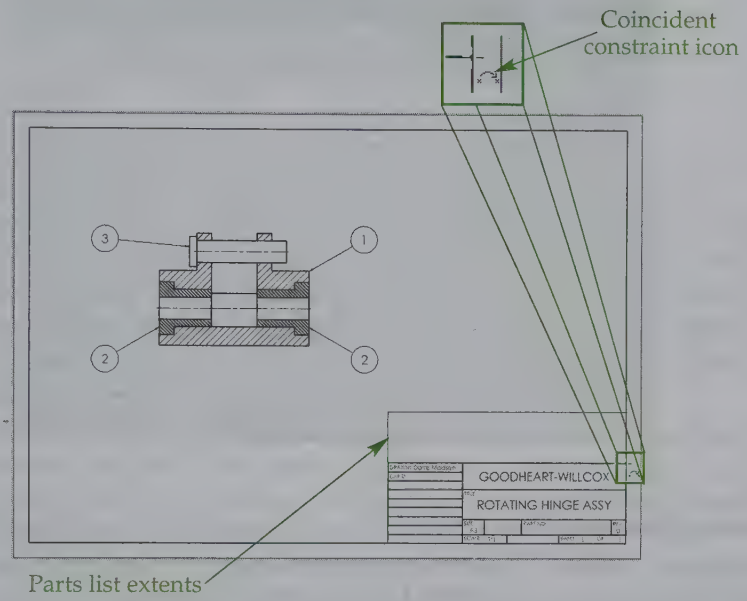
Once you select an assembly view or file, you are ready to generate the parts list. However, you can specify additional parts list options using the **BOM Settings and Properties** and **Table Wrapping** areas. The **BOM Settings and Properties** area includes the same settings available in the **BOM Properties** dialog box. The **Table Wrapping** area includes options used to identify the parts list layout. For assembly drawings with large parts lists, you can pick **Enable Automatic Wrap** check box to divide the parts list into multiple section. When you pick the **Enable Automatic Wrap** check box the **Maximum Rows** and **Number of Selections** radio buttons become available for selection. Choosing the **Maximum Rows** radio button splits the parts list when the number of rows defined in the **Maximum Rows** edit box is greater than the number of rows in the parts list. For example, if a parts list contains 30 items, and the **Maximum Rows** edit box is set to 10, the parts list is split into three sections of ten components, or rows. Choosing the **Number of Sections** radio button splits the parts list into the number of sections defined in the **Number of Sections** edit box. As a result, the parts list items, or components, are divided among the specified number of columns. So far, the parts lists shown in this chapter contain only one section. **Figure 19-24** shows an example of a parts list with two sections. Once you define the number of sections, pick either the **Left** or **Right** radio button to specify where multiple table sections are placed in reference to each other. A left table wrapping, shown in **Figure 19-24**, creates multiple sections from right to left, while a right table wrapping creates multiple sections from left to right.

Once you select a drawing view or file and define additional parts list display characteristics, pick the **OK** button on the **Parts List** dialog box to insert the parts list. A rectangle that represents the parts list extents is shown and helps you place the parts list on the sheet. See **Figure 19-25**. Use your cursor to select a point where you want the parts list to be placed, and to generate the parts list.

Figure 19-24.
An example of a parts list with two sections.

				3	1	RHP 65743967	PIN
1	1	RHH 9567	HOUSING	2	2	RHB 4582364	BUSHING
ITEM	QTY	PART NUMBER	DESCRIPTION	ITEM	QTY	PART NUMBER	DESCRIPTION
PARTS LIST				PARTS LIST			

Figure 19-25.
Select an existing object, such as a title block, when placing a parts list.



PROFESSIONAL TIP



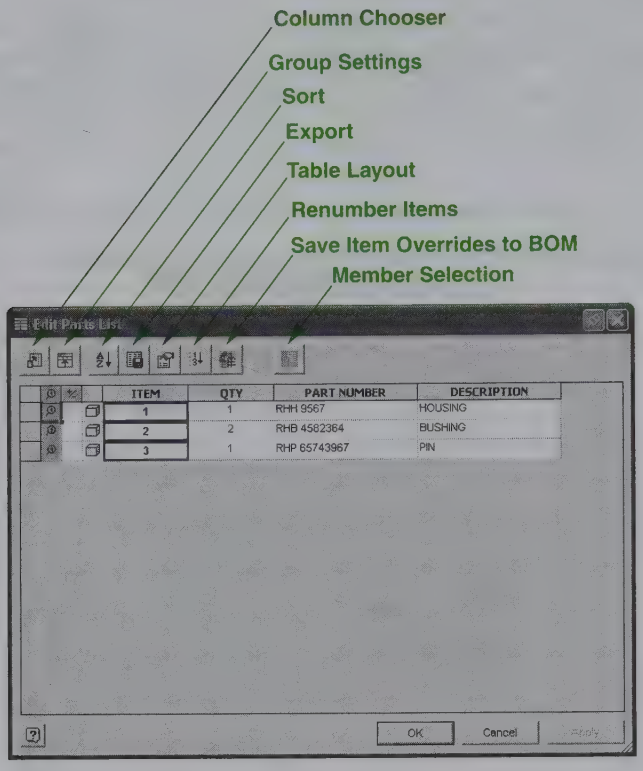
Automatic collinear and coincident constraining should help to position the parts list according to the border and title block. The collinear and coincident icons identify when the constraint is made, in addition to a noticeable snap.

Modifying Parts Lists

Once you create a parts list, you can edit much of its information and position. A parts list can be moved like other Inventor items. When you select the parts list, green buttons are displayed at each corner. Use these buttons to move the parts list to a desired location either in space or by linking a corner of the parts list to an existing drawing point, such as the corner of a border or title block. You can also use the green indicator buttons to increase or decrease the size of the parts list.

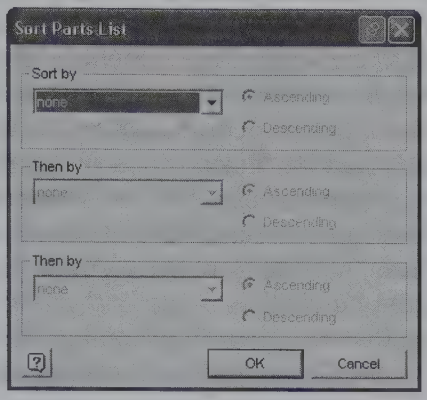
Most parts list editing options are controlled in the **Edit Parts List** dialog box, shown in **Figure 19-26**, available by right-clicking on a parts list and selecting the **Edit Parts List...** pop-up menu option or double-clicking on the parts list. The following items are available in the **Edit Parts List** dialog box and allow you to override the default display characteristics of an existing parts list:

Figure 19-26.
The **Edit Parts List** dialog box.



- **Column Chooser.** Select this button to open the **Parts List Column Chooser** dialog box, which is used to add columns to the parts list by referencing specific properties. The **Parts List Column Chooser** dialog box is the same **Parts List Column Chooser** dialog box available from the **Parts List Style** area of the **Style and Standards Editor**.
- **Group Settings.** Pick this button to access the **Group Settings** dialog box, which allows you to control the display of separate assembly drawing components that have the same properties, in the parts list. The **Group Settings** dialog box is the same **Parts List Column Chooser** dialog box available from the **Parts List Style** area of the **Style and Standards Editor**.
- **Sort.** When you select this button, the **Sort Parts List** dialog box is displayed. See Figure 19-27. The **Sort Parts List** dialog box allows you to modify the current parts list order. Select the property you want to sort first by picking a property from the **Sort by** drop-down list. Then specify whether you want the property displayed in ascending order or descending order by selecting either the **Ascending** or **Descending** radio button. Next, if required, identify which property you want to sort second, when the first property is not available, by picking a property from the **Then by** drop-down list. Specify whether you want the property displayed in ascending or descending order by selecting either the **Ascending** or **Descending** radio button. Finally, if needed, identify which property you want to sort third, when the first and second properties are not available, by selecting a property from the second **Then by** drop-down list. Again, specify whether you want the property displayed in ascending or descending order by selecting either the **Ascending** or **Descending** radio button.
- **Export.** Pick this button if you want to export, or save, the parts list to an external, or non-Inventor type of file. When you select the **Export** button, the **Export Parts List** dialog box is displayed. See Figure 19-28. The **Export Parts List** dialog box

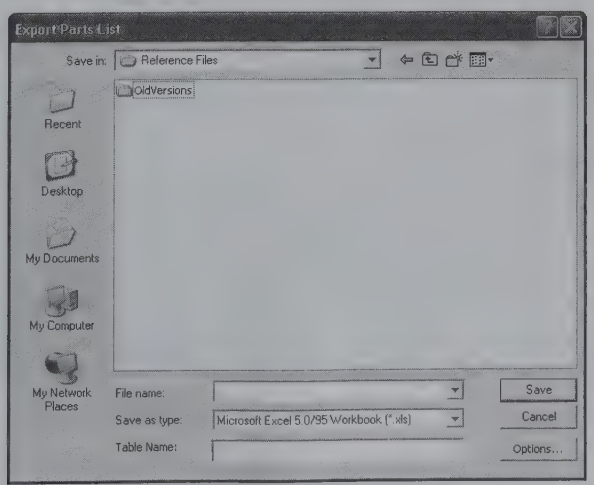
Figure 19-27.
The **Sort Parts List** dialog box.



functions much like any other **Save As** dialog box. However, in addition you can enter the name of the parts list table in the **Table Name** edit box.

- **Table Layout.** Choose this button to open the **Parts List Table Layout** dialog box, which allows you to override a variety of parts list layout characteristics. The options available in the **Parts List Table Layout** dialog box are the same as those in the **Parts List Style** area of the **Style and Standards Editor**. The only difference is that you cannot modify the heading and row gaps using the **Parts List Table Layout** dialog box.
- **Renumber.** This button renumbers the parts list columns consecutively, as defined in the parts list standards. Pick the **Renumber** button to override the previously described sort options, value edits, or other numerical modifications, in order to renumber the parts list.
- **Parts List Spreadsheet.** The parts list spreadsheet, or table, displays parts list information and allows you to modify parts list values. Many of the options associated with the parts list spreadsheet have already been described. However, the spreadsheet also contains many other parts list manipulation options. The balloon symbols identify which parts list items have corresponding balloons in the assembly drawing. Also, you can edit any of the information in the cells by selecting inside the cell, or by highlighting and redefining the existing value.

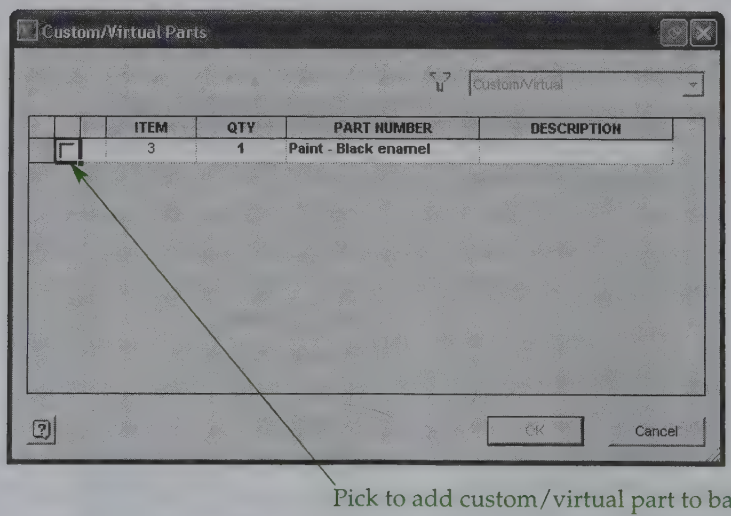
Figure 19-28.
The **Export Parts List** dialog box.



Column width can also be edited by selecting the line between two columns and dragging the column to the desired width. In addition, if you right-click on any of the spreadsheet cells, the following menu options are listed:

- **Visible.** When this option is selected, the row is visible. Conversely, deselect the **Visible** option to turn off, or hide, the row display.
- **Wrap Table at Row.** Select this option to split, or divide, the column at the selected row. This option functions similar to the **Enable Automatic Wrap** check box in the **Parts List** dialog box. For assembly drawings with large parts lists, you may want to split columns so that the assembly components are divided among a specified number of columns.
- **Insert Custom Part.** Pick this option to place a new custom row in the parts list below the currently selected table row. A custom part and the corresponding custom **Edit Parts List** dialog box spreadsheet row can be added anytime you want to include additional custom parts list information. Once you place a custom parts list row, you can create a corresponding custom balloon using the **Balloon** tool, or attach a custom balloon to an existing balloon. To create a custom balloon, follow the steps previously described for adding balloons, but after you pick a start position and before you generate the balloon, right-click and select the **Custom/Virtual** pop-up menu option. Then, right-click and pick the **Continue** pop-up menu option to display the **Custom/Virtual Parts** dialog box. See Figure 19-29. Select the desired custom part row, and pick the **OK** button to generate the custom balloon. To add a custom balloon to an existing balloon, right-click on an existing balloon, and pick the **Attach Custom/Virtual Balloon** pop-up menu option. Then select the check box at the beginning of the desired custom row from the **Custom/Virtual Parts** dialog box. Pick the **OK** button, and choose a location for the additional balloon.
- **Remove Custom Part.** Select this option to remove a custom part row.
- **Save Item Overrides to BOM.** Pick this option if you have made changes to any of the values in the parts list spreadsheet and you want the changes saved to the assembly model bill of materials.
- **Static Value.** Pick this option to make the selected parts list table value static, so it does not update when the drawing view or parent file is updated.

Figure 19-29.
The **Custom/Virtual Parts** dialog box.



- **Column properties.** Right-clicking on a column presents options that allow you to override a number of parts list column properties. Select the **Table Layout...** option to access the **Parts List Table Layout** dialog box previously described. Pick the **Column Chooser...** option to open the **Parts List Column Chooser** dialog box. Pick the **Group Settings** button to access the **Group Settings** dialog box. Select the **Format Column...** option to access the **Format Column** dialog box specific to the selected column. Use the **Format Column** dialog box to override the column format characteristics specified in the **Parts List Style** area of the **Style and Standards Editor**. And finally, pick the **Column Width...** option to open the **Column Width** dialog box specific to the selected column. Use the **Column Width** dialog box to override the width of the column specified in the **Parts List Style** area of the **Style and Standards Editor**.

Once you make all necessary parts list modifications, pick the **OK** button to generate the changes and exit the **Edit Parts List** dialog box.

NOTE



Although the properties of a single parts list can be modified using the **Edit Parts List** dialog box, to set the default parts list standards, use the **Parts List Style** area of the **Style and Standards Editor** detailed earlier in this book.



Complete the Exercise on the Student CD.

Using Weldment Model Symbols and Annotations

Most 2D drawings, including weldment drawings, are generated from existing 3D models. As a result, both weldment model and drawing symbols and annotations are available in Inventor. Weldment model symbols and annotations are specified when you add weldment information to the weldment model, including weld beads and weld symbols. When you insert a weldment model into a drawing, the model symbols and annotations can be automatically displayed. See **Figure 19-30**. Unlike other drawing model dimensions, weldment model symbols and annotations are not fully parametric, and cannot be edited to change the physical parameters of the corresponding weldment model weld beads or symbols.

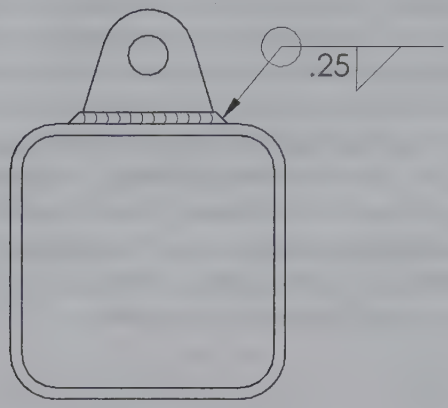
NOTE



Weldment model symbols can be displayed in addition to model dimensions.

Weldment model annotations are the weld beads created using the **Fillet Weld** and **Groove Weld** tools, explained earlier in this book. Cosmetic welds do not allow weldment model annotation geometry to be displayed in a drawing view. Weldment model symbols are the weld symbols created using the **Fillet Weld**, **Cosmetic Weld**, and **Groove Weld** tools. Only weldment model symbols that are planar to the drawing view can be shown. Use one of the following techniques to display weldment model annotations (weld bead geometry):

Figure 19-30.
This is an example of a weldment drawing view with model symbols and annotations displayed.



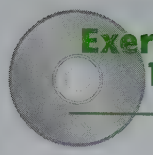
- When placing a base view using the **Base View** tool, select the **Weld Annotations** check box from the **Options** tab of the **Drawing View** dialog box.
- Edit a drawing view, and select the **Weld Annotations** check box from the **Options** tab of the **Drawing View** dialog box.
- Right-click inside an existing drawing view and select the **Get Weld Annotations** option from the **Get Model Annotations** cascading submenu.
- Right-click on an existing drawing view name in the browser, and select the **Get Weld Annotations** option from the **Get Model Annotations** cascading submenu.

Use one of the following techniques to display weldment model weld symbols:

- When placing a base view using the **Base View** tool, select the **Model Weld Symbols** check box from the **Options** tab of the **Drawing View** dialog box.
- Edit a drawing view, and select the **Model Welding Symbols** check box from the **Options** tab of the **Drawing View** dialog box.
- Right-click inside an existing drawing view and select the **Get Welding Symbols** option from the **Get Model Annotations** cascading submenu.
- Right-click on an existing drawing view name in the browser, and select the **Get Welding Symbols** option from the **Get Model Annotations** cascading submenu.

Typically, weldment model weld symbols are placed directly over view geometry. You can move weld symbols by selecting the appropriate green indicator button, and dragging the entire dimension to the desired location. You can never actually remove a weldment model weld symbol from a drawing, unless you delete the parameter from the model, in the assembly file. However, you can hide individual weld symbols by right-clicking on the symbol you want to hide and selecting the **Delete** pop-up menu option. You can also hide all model dimensions in a view by right-clicking inside a view and deselecting the **Model Welding Symbols** option from the **Annotation Visibility** cascading submenu. Similarly, weldment annotations can be hidden by right-clicking on the weld bead you want to hide and selecting **Delete** or deselecting the **Visibility** pop-up menu option. You can also hide all weldment annotations in a view by right-clicking inside a view and deselecting the **Model Annotations** option from the **Annotation Visibility** cascading submenu.

If you want to change the size or shape of a weldment annotation while working in the drawing environment, right-click the weld bead you want to change, and select the **Edit Caterpillar...** pop-up menu option. When you pick the **Edit Caterpillar...** option, the **Weld Caterpillars** dialog box, explained later in this chapter, is displayed.



Complete the Exercise on the Student CD.

Placing End Fills

When you add weld beads to a weldment model, the weld bead geometry is displayed in the drawing view, assuming you display at least the welds stage of weldment design. However, if a model does not include weld beads, or if you need to add a weld representation, you can add end fills in the drawing environment. An ***end fill*** is a representation of a weld bead end shown on the edge of a weldment. End fills are not associated with model welds and are only created in the drawing environment. See Figure 19-31.

To add an end fill, access the **End Fill** tool by selecting the **End Fill** flyout button from the **Caterpillar** button on the **Drawing Annotation Panel** bar, or selecting the **End Fill** flyout button from the **Caterpillar** button on the **Drawing Annotation Panel** toolbar.

When you access the **End Fill** tool, the **End Fill** dialog box is displayed. See Figure 19-32. To create an end fill, first define the type of end fill geometry you want to place by selecting the desired button from the **Type** tab. The **Preset Shape** area contains preset end fill type buttons including **Fillet**, **J-Type**, **V-Type**, **U-Type**, and **Seam**. When you pick a **Present Shape** area button, size options are displayed in the **Size** area of the **Type** tab. Use the edit boxes that correspond to the graphic to define the size of the end fill. The edit boxes are used to specify the weld leg sizes for fillet weld end fills. The edit boxes define the weld width and depth for J-type, V-type, and U-type weld end fills. The edit box determines the height of the weld and the width is specified by selecting two points for seam end fills. An alternative end fill shape option is to create a sketched region by picking the **Custom Shape** button in the **Custom Shape** area. Sketching a custom region is just like sketching a profile using the **Line** tool in the 2D sketch work environment.

Once you identify the type of end fill you want to display, pick the **Options** tab, shown in Figure 19-33, and use the options in the **Fill** area to define the end fill display characteristics. By default, the **Solid Fill** check box is selected, allowing you to add a solid color to the end fill region. If you do not want to add a solid color fill, deselect the **Solid Fill** check box and choose a hatch pattern from the **Hatch** drop-down list. Then, specify the hatch pattern scale by entering a scale in the **Scale** edit box, or pick the flyout button

Figure 19-31.
An example of a weldment drawing end fill, a fillet end fill.

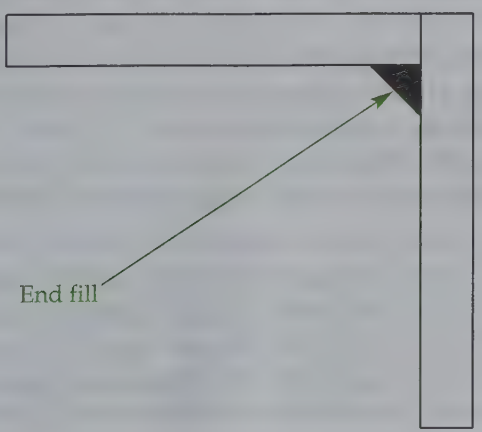


Figure 19-32.
The **End Fill** dialog box, **Type** tab.

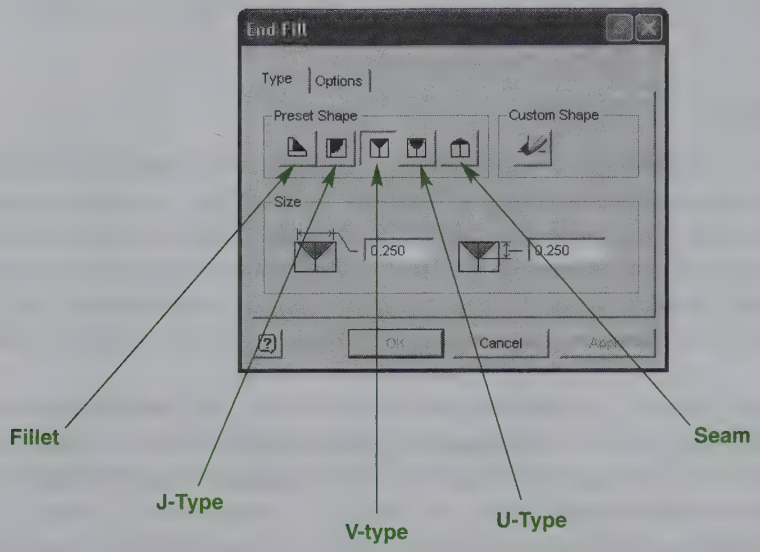
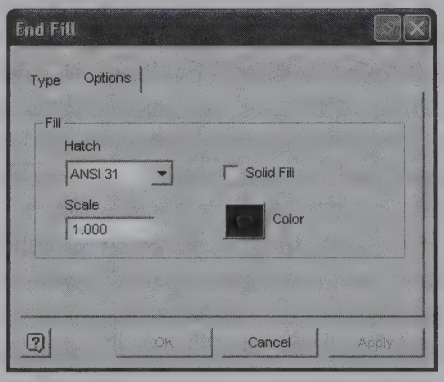


Figure 19-33.
The **End Fill** dialog box, **Options** tab.



and select a scale. Finally, identify the fill color by selecting the **Color** button and choosing a color from the **Color** dialog box. In most situations, the **Color** variable should be set to By Layer, which will automatically assign a color to the end fill based on the characteristics of the layer defined in the **Object Defaults Style** area.

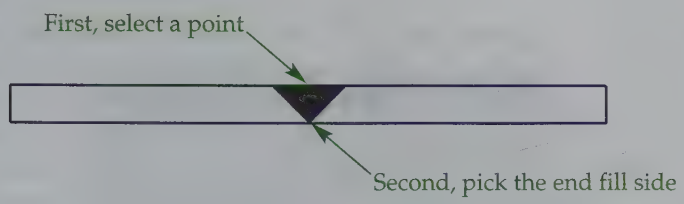
When you fully define all the **End Fill** dialog box settings, pick a point on the drawing view to place the end fill, and select the side on which the end fill is added. See Figure 19-34. When you are satisfied with the end fill placement, pick the **OK** button on the **End Fill** dialog box to generate the geometry and exit the **End Fill** tool. Or, right-click and pick the **Apply** pop-up menu option, or select the **Apply** button on the **End Fill** dialog box. The **End Fill** tool remains open, allowing you to place additional end fills. When finished, pick the **Cancel** button to close the **End Fill** dialog box.

NOTE



The process for adding end fills varies depending on the selected shape, though it still involves picking one or two points, followed by the side on which the end fill is placed in reference to the initial selection or selections.

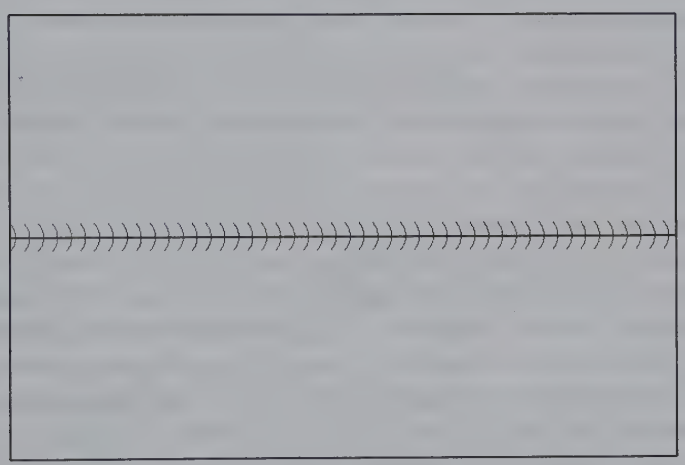
Figure 19-34.
Creating an end fill, a V-type end fill.



Adding Caterpillars

When you add weld beads to a weldment model, the welds can be shown in the drawing view if you display at least the welds stage of weldment design, and weld annotations are visible. However, if required, you can add caterpillars to weldment drawing views. A ***caterpillar*** is a representation of a weld bead and is not related to a weldment model weld. See **Figure 19-35**.

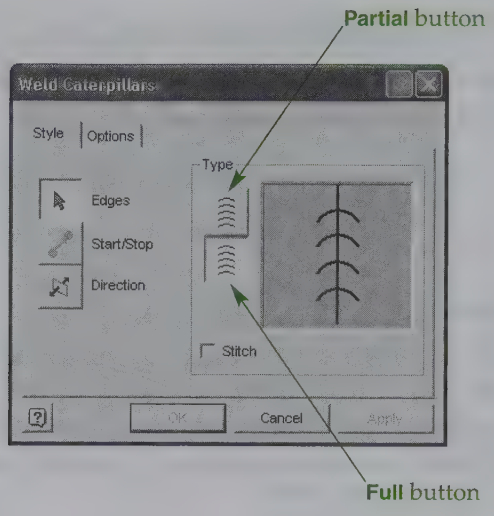
Figure 19-35.
An example of a caterpillar found on a weldment drawing.



To add a caterpillar, access the **Caterpillar** tool by selecting the **Caterpillar** flyout button from the **Drawing Annotation Panel** bar, or selecting the **Caterpillar** flyout button from the **Drawing Annotation Panel** toolbar.

When you access the **Caterpillar** tool, the **Weld Caterpillars** dialog box is displayed. See **Figure 19-36**. To create a caterpillar, first define the type of caterpillar you want to place, by selecting the desired button from the **Type** area. The **Type** area contains a **Partial** button that allows you to place a caterpillar next to an edge, such as with a fillet weld, and a **Full** button that allows you to place a caterpillar along both sides of an edge, such as with a V- or square-groove weld. The **Type** area also contains the **Stitch** check box, which allows you to define an intermittent weld. The next step to adding a caterpillar is to pick the edge on which to place the weld bead, assuming the **Edge** button is selected by default. Then, if required, pick the **Start/Stop** button and select a point where the weld begins and a point where the weld ends. You can also redefine the weld flow direction by selecting the **Direction** button.

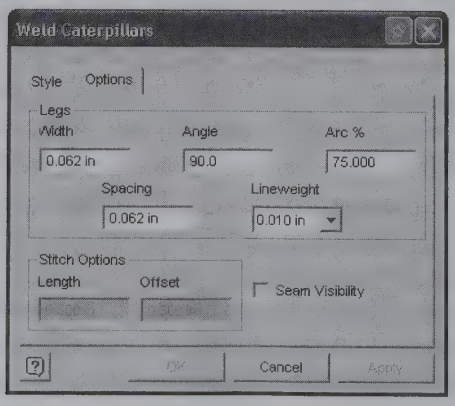
Figure 19-36.
The **Weld Caterpillars** dialog box, **Style** tab.



Once you identify the caterpillar type and placement, pick the **Options** tab, shown in Figure 19-37, and use the following options to define end fill display characteristics:

- **Legs.** This area allows you to specify the caterpillar size, and includes the following options:
  - **Width.** Use this edit box to specify the weld bead width.
  - **Angle.** This edit box is used to define the angle between the selected edge and the caterpillar geometry. For example, you might use a 90° angle for a square groove weld and a 45° angle for a certain type of fillet weld.
  - **Arc %.** This edit box is used to specify the radius of the caterpillar arc in reference to the arc chord, or bead width.
  - **Spacing.** Use this edit box to define the spacing between caterpillar arcs.
  - **Lineweight.** Use this drop-down list to select the desired caterpillar lineweight. In most situations, the **Lineweight** variable should be set to By Layer, which will automatically assign a lineweight to the caterpillar based on the characteristics of the layer defined in the **Object Defaults Style** area.
- **Stitch Options.** This area is available if the **Stitch** check box of the **Style** tab is selected, and allows you to create an intermittent weld bead. Enter the length of each weld bead in the **Length** edit box. Then enter the center-to-center

Figure 19-37.
The **Weld Caterpillars** dialog box, **Options** tab.



distance between weld lengths, or pitch, in the **Offset** edit box. For example, a caterpillar with a weld length of 25 mm and an offset of 45 mm has 20 mm space between weld beads.

- **Seam Visibility.** Pick this check box to display the edge on which the caterpillar is placed once the caterpillar is generated.

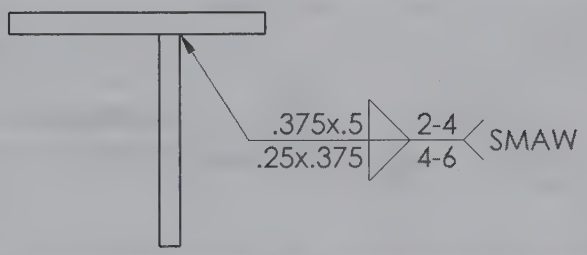
When you fully define all the **Weld Caterpillars** dialog box options, pick the **OK** button to generate the caterpillar and exit the **Caterpillar** tool. Or, right-click and pick the **Apply** pop-up menu option, or select the **Apply** button on the **Weld Caterpillars** dialog box. The **Caterpillar** tool remains open, allowing you to place additional caterpillars. When finished, pick the **Cancel** button to close the **Weld Caterpillars** dialog box.

Placing Weld Symbols

Weld symbols can be included when you add welds to a weldment model using any of the weld tools. Model weld symbols are shown in drawing views, assuming you display at least the welds stage of weldment design, and weld symbols are visible. However, if required, you can add additional weld symbols to weldment drawing views, or to drawings that are not created using weldment models. 2D drawing weld symbols are not related to weldment models, but appear similar. See **Figure 19-38.**

To create drawing weld symbols in the drawing work environment, access the **Welding Symbol** tool by picking the **Welding Symbol** button on the **Drawing Annotation Panel** bar, or picking the **Welding Symbol** button on the **Drawing Annotation Panel** toolbar.

Figure 19-38.
This is an example of a weldment drawing weld symbol.



Once you access the **Welding Symbol** tool, use your cursor to select a feature or point where you want to add welding information. Then drag the weld symbol to the desired location and pick. See **Figure 19-39.** After you select appropriate drawing geometry and locate the weld symbol, press the [Enter] key or right-click and select the **Continue** menu option, to display the **Welding Symbol** dialog box. See **Figure 19-40.**

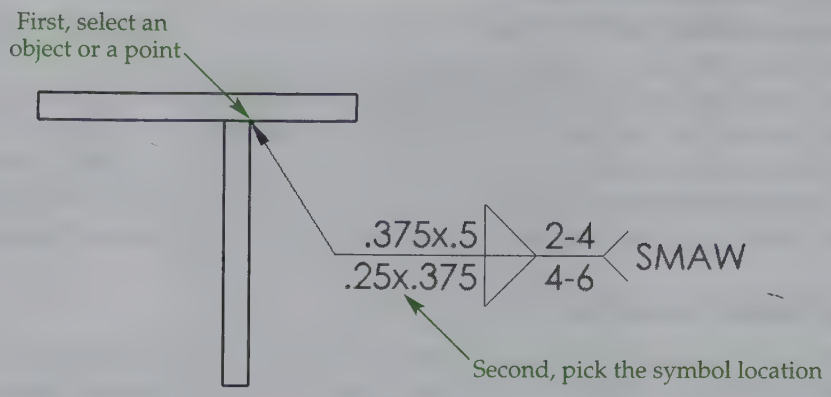
NOTE

The options in the **Welding Symbol** dialog box will vary depending on the settings you specify in the **Weld Symbol Style** area of the **Style and Standards Editor.**

The following options are available in the **Welding Symbol** dialog box:

- **Identification Line.** This flyout button is used to control the placement of the identification line, if used. An ***identification line*** is placed in addition to the reference line primarily when using BSI and certain other drafting standards. The ***reference line*** is the line on which weld symbol information is displayed. By default, the **No Identification Line** button is selected, and an identification

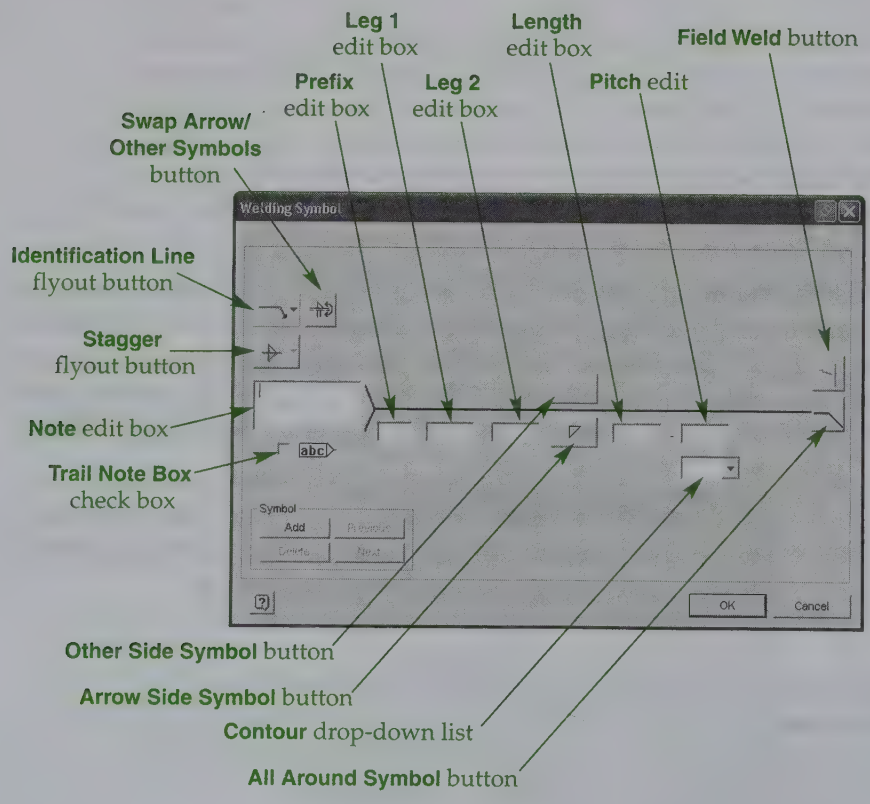
Figure 19-39.
Placing a welding symbol.



line is not shown. Select the **Identification Line - Arrow Side** or **Identification Line - Other Side** option from the flyout, depending on the application. The effects of the selected identification line button are displayed when the weld is created and weld symbol is displayed.

- **Swap Arrow/Other Symbols.** Use this button to define the selected side on which a weld bead is placed according to the welding symbol. For example, if you are currently defining an arrow side weld, when you pick the deselected **Swap Arrow/Other Side** button, you specify the other side instead of the arrow side.

Figure 19-40.
The **Welding Symbol** dialog box.



- **Stagger.** This flyout button is only available if you specify an other-side symbol using the **Other Side** button. Use the **Stagger** flyout button to define the orientation of weld symbols. By default, the button is set to **No Stagger**, but you can choose from a variety of stagger options depending on the application and standard. Again, the effects of the selection button are displayed when the weld is created and weld symbol is shown.
- **Note.** This edit box is used to add a weld note to the symbol. To display a note, type the required information in the edit box.
- **Trail Note Box.** Depending on the specified drafting standard, choose this check box to add a box around the weld note, if you added a note to the **Note** edit box.
- **Prefix.** Use this edit box to define a weld symbol prefix. A *prefix* is a word, character, or symbol added before the weld size.
- **Leg 1.** Use this edit box to identify the length of the first weld bead leg in the welding symbol.
- **Leg 2.** Use this edit box to identify the length of the second weld bead leg in the welding symbol.
- **Arrow Side Symbol.** This button is used to describe the type of weld you create on the arrow side of the joint. If you are not placing the default fillet weld, pick the **Arrow Side Symbol** button and choose a different symbol.
- **Length.** Enter the length of an intermittent weld in this edit box.
- **Pitch.** Enter the pitch of an intermittent weld in this edit box.
- **Contour.** Select a contour from this drop-down list.
- **Other Side Symbol.** By default, an arrow-side welding symbol is created, unless you pick the **Swap Arrow/Other Symbols** button previously described. In order to use a symbol with information regarding both sides, you must pick the **Other Side Symbol** button or the **Arrow Side Symbol** button, if the **Swap Arrow/Other Symbols** button is active, and select a symbol from the palette. When you pick a symbol, the other-side options become available. The other-side preferences available are the same as the arrow-side options previously described, but are located above the weld symbol reference line.
- **Field Weld.** Pick this button to add a field weld symbol for field welding applications.
- **All Around.** Select this button to add a weld all-around symbol, for applications that require all-around welding.
- **Symbol.** This area is used to add an additional symbol reference line to the weld symbol leader. Typically, another reference line is added after the initial line has been fully defined. However, you can add another symbol anytime by selecting the **Add** button. Then, pick the **Next** button to activate the added symbol. Once activated, you can adjust the settings of the added reference line. You can also return to a previous reference line by selecting the **Previous** button, or delete a reference line by picking the **Delete** button.

When you fully define the weld symbol specifications in the **Welding Symbol** dialog box, pick the **OK** button to create the weld symbol. You will notice the **Welding Symbol** tool is still active. Continue placing weld symbols as required or right-click and select the **Done** menu option, press [Esc], or access a different tool.



Exercise 19-7 Complete the Exercise on the Student CD.

Multiple Sheet Drawings

A set of working drawings typically contains a multiple number of sheets. Generally, a fully defined assembly or weldment drawing, with a parts list, is printed on one sheet, and each detail drawing is printed on a separate sheet. For example, five separate sheets of paper may be used to create a set of working drawings for a four-part assembly. Individual Inventor drawing sheets are used in the same way that individual pieces of paper are used to document an assembly and each of the assembly components. This does not mean that you have to open several separate drawing files, because each working drawing sheet is contained in one drawing file. You should be able to document an entire design with one drawing file and multiple drawing sheets.

A set of working drawings is developed by adding the assembly or weldment drawing and detail drawings together in one drawing file. The process of generating multiple sheet drawings can be accomplished using a variety of techniques. Probably the most effective way of creating multiple sheet drawings is to open a new drawing file, create the assembly drawing, and then create the detail drawings. However, you may choose any approach you find effective. You can even insert existing sheets from a drawing file into the multiple drawing sheet file.

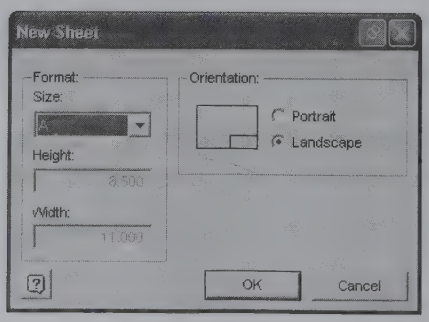
Creating a New Blank Sheet

To create a new blank sheet, select **Sheet...** from the **Insert** pull-down menu, or press [Shift]+[N]. When you select this option, the **New Sheet** dialog box is displayed, as shown in **Figure 19-41**. A new blank sheet does not have any sheet specifications, a border, a title block, or any other information. The **New Sheet** dialog box is similar to the **Edit Sheet** dialog box described earlier in this book, and allows you to define the new sheet parameters. The following options are available inside the **New Sheet** dialog box:

- **Size.** This drop-down list allows you to specify the sheet size. There are a number of standard sheet sizes available such as A, B, C, D, E, and F. You can also define your own sheet size by selecting the **Custom Size** option for the desired units (**Custom Size [inches]** or **Custom Size [mm]**). Once you choose the custom size option, the **Height** and **Width** edit boxes become active, allowing you to enter the desired height and width of the custom sheet size.
- **Portrait.** This radio button allows you to apply a portrait, or vertical, sheet orientation.
- **Landscape.** This radio button allows you to apply a landscape, or horizontal, sheet orientation.

Once you have defined the desired sheet options, pick the **OK** button to generate the sheet.

Figure 19-41.
The **New Sheet** dialog box.



Reusing an Existing Sheet

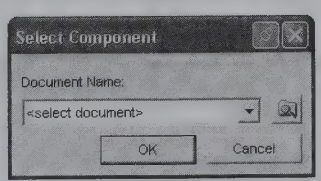
To create a new sheet with the same layout as an existing sheet, right-click in the graphics window background and select the **New Sheet** pop-up menu option; right-click in the browser and select the **New Sheet** pop-up menu option; pick the **New Sheet** button on the **Drawing Annotation Panel** bar; or pick the **New Sheet** button from the **Drawing Annotation Panel** toolbar. When you use this sheet creation option, a copy of the existing sheet is added, complete with the same sheet parameters, border, and title block as the existing sheet. If you do not want to use the same title block, delete the title block or select the **Undo** button until the title block is removed. Similarly, if you do not want to use the same border, delete the border or select the **Undo** button until the border is removed.

An existing sheet in a separate drawing file can be copied and pasted into the current drawing as a new sheet. Copying a sheet brings all information, including drawing views and annotations, from one file to another. To copy an existing drawing sheet from a drawing file into another file, first open the drawing file where the drawing sheet is located. Then, right-click on the drawing sheet in the graphics window, or the sheet name in the browser, and select the **Copy** menu option. Or, select the sheet and press [Ctrl]+[C]. Next, with the original, or source, file still open, access the drawing file where you want to place the copy, and right-click on the graphics window background or the drawing file name in the browser, and select the **Paste** menu option, or select the sheet and press [Ctrl]+[V].

Using a Sheet Format

A ***sheet format*** is a template stored in the drawing file. When you use a sheet format, a new sheet is created, and views from a specified model are inserted onto the sheet along with a border and title block, assuming they are part of the sheet format. To use an existing sheet format, expand the **Drawing Resources** folder and the **Sheet Formats** folder in the browser. Then, double-click the sheet format you want to use, or right-click the sheet format and select the **New Sheet** menu option. Once the sheet format is selected, the **Select Component** dialog box appears. See **Figure 19-42**. If you previously inserted a component, the model is available in the **Document Name** drop-down list. Otherwise, you must pick the **Browse** button to display the **Open** dialog box, and locate the model you want to insert. Once you locate the file, pick the **OK** button to create the sheet, complete with a border, title block, and the specified views.

Figure 19-42.
The **Select Component** dialog box.



Working with Multiple Sheets

Once a new sheet is added to a drawing file, you can use any of the tools and techniques explained in this chapter and previous chapters to create a complete set of working drawings. In addition to the tools and drawing processes described, you may want to become familiar with some concepts related specifically to multiple sheet drawings, including numbering each sheet, changing the order of sheets, and moving drawing views between sheets.

Figure 19-43 shows the browser display of a drawing file with five drawing sheets. The contents of the active sheet are displayed in the graphics window. Only one sheet can be active at a time. The active sheet in Figure 19-43 is Sheet:1; all other sheets are shaded. To activate a different sheet, double-click the sheet name or right-click on the sheet name and select the **Activate** menu option. The sheet number or page number is identified in the browser by drawing sheet order. As in other Inventor applications, you can drag a sheet above or below another sheet to change the order.

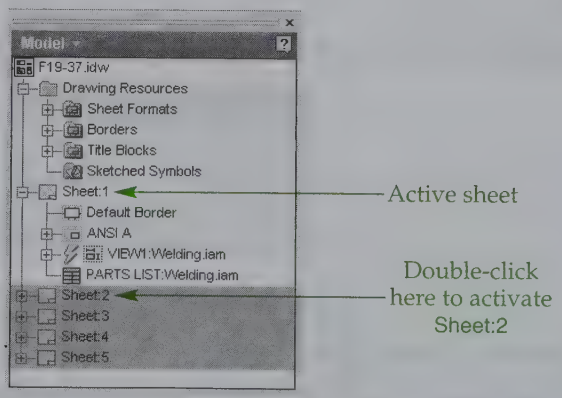
The order of sheets in the browser also defines the drawing title block information, when drawing properties are used. For example, if there are five sheets in the drawing file, and each sheet has a title block with the **Sheet Number** and **Number of Sheets** drawing property fields, the first sheet in the browser is sheet 1 of 5, while the last sheet in the browser is sheet 5 of 5. In addition, title block **Properties - Model** corresponds to the model in the drawing view while **Properties - Drawing** corresponds to the properties defined in the **Properties** dialog box for the drawing file in which you are currently working.

Although you cannot copy a sheet within the same drawing file, in some situations you may want to copy or move a view from one sheet to another. Copying a view is similar to copying an entire sheet. First, right-click on the drawing view in the browser, or open the sheet with the desired view, right-click on the drawing view, and select the **Copy** menu option. Or, select the view and press [Ctrl]+[C]. Next, right-click on the drawing sheet in the browser where you want to place the view, and select the **Paste** menu option. Or, select the sheet and press [Ctrl]+[V]. To move a view from one sheet to another, drag the view name from its current location in the browser to the new sheet location in the browser. When you move a base or dependent view from one sheet to another, a shortcut is created. The shortcut and the shortcut icon in the browser identify that there is a relationship between two views in different sheets. To view the relationship, right-click on the shortcut icon, and select the **Go To...** menu option to open the corresponding sheet.



Exercise 19-8 Complete the Exercise on the Student CD.

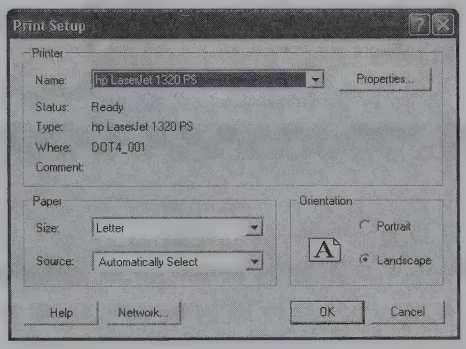
Figure 19-43.
An example of the browser display for a multiple-sheet drawing.



Printing Drawings

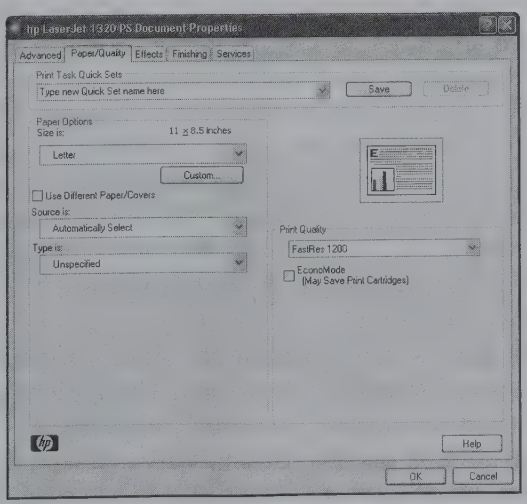
Once you create your drawings, you will probably want to print, or plot, the drawings onto a sheet of paper. It is crucial that a drawing is printed correctly, and to scale, when required. In order to print a drawing, you may first want to define print specifications by selecting **Print Setup...** from the **File** pull-down menu. Selecting the **Print Setup** option opens the **Print Setup** dialog box shown in **Figure 19-44**. The following options are available inside the **Print Setup** dialog box:

Figure 19-44.
The **Print Setup** dialog box.



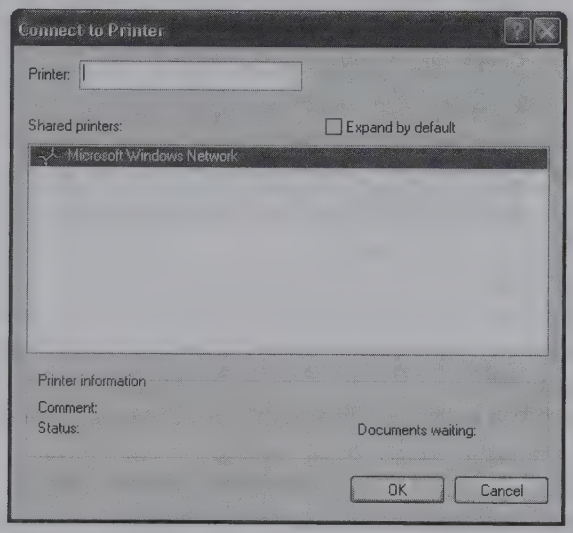
- **Name.** This drop-down list allows you to select the device you want to use to plot your document.
- **Status.** Displays the status of the print job, for example, printing, canceled or complete.
- **Type.** Displays the type of printer currently being used.
- **Where.** Displays the printer port currently being used by the specified printer.
- **Properties.** Selecting this button opens the **Properties** dialog box for the specified printer. See **Figure 19-45**. Here you can adjust the page layout, paper source, print quality, and other options that apply to the specified print device.

Figure 19-45.
An example of a plot device **Properties** dialog box.



- **Size.** This drop-down list lets you select the size of paper to use for the document.
- **Source.** This drop-down list lets you select from where the paper comes.
- **Orientation.** Inside this area of the **Print Setup** dialog box, you can select the **Portrait** radio button or the **Landscape** radio button. The **Portrait** radio button allows you to rotate and print the document in a portrait, or vertical, orientation. The **Landscape** radio button allows you to rotate and print the document in a landscape, or horizontal, orientation.
- **Network.** Selecting this button opens the **Connect to Printer** dialog box, shown in Figure 19-46. If you are working in a networked setting and are not connected to a local printer, use this dialog box to connect to a shared network printer and plot to that device.

Figure 19-46.
The **Connect to Printer** dialog box.

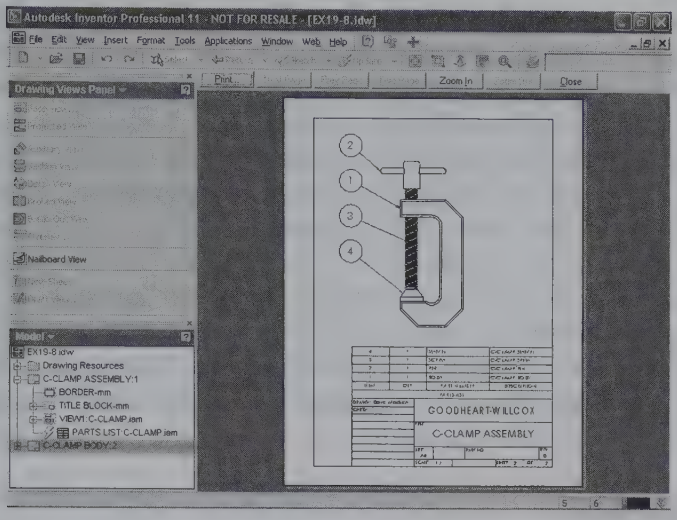


To preview the intended print job using the current print specifications, access the **Print Preview** dialog box, shown in Figure 19-47, by selecting **Print Preview** from the **File** pull-down menu. From the print preview area, you can initiate the print by selecting the **Print...** button, review other pages that will print with the current print job by selecting **Next Page**, **Prev Page**, or **Two Page** buttons, and zoom in or zoom out to further review the print preview by choosing the **Zoom In** or **Zoom Out** buttons. Then select the **Close** button to terminate the print preview.

The **Print Setup** dialog box is useful for specifying print settings, and the **Print Preview** dialog box is effective for previewing and printing a drawing. However, usually you must define additional drawing print specifications, such as scale, in order to print the drawing correctly. To define printing parameters and print a drawing, access the **Print Drawing** dialog box, shown in Figure 19-48, by selecting **Print...** from the **File** pull-down menu. As you can see, the **Print** dialog box is the most powerful printing tool and allows you to set up, preview, and print drawings. The following options are available inside the **Print** dialog box:

- **Printer.** This area allows you to define the device to use for the print job. Use the **Name** drop-down list to select a printer. Then, pick the **Properties** button to access the **Properties** dialog box previously explained.
- **Print Range.** This area is used to define which sheets in the drawing file you want to print. Select the **Current Sheet** radio button if you want to print only

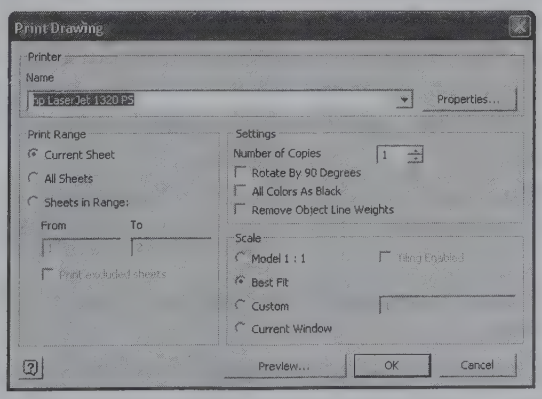
Figure 19-47.
A preview of a print job.



the active drawing sheet, or select the **All Sheets** radio button to print all the sheets in the drawing file. You can also choose to print a certain range of sheets by picking the **Sheets in Range** radio button and entering the sheet you want to print first in the **From** edit box, and the sheet you want to print last in the **To** edit box. If you want to override the **Exclude From Print** option specified in the **Edit Sheet** dialog box and print the sheets specified as excluded, select the **Print excluded sheets** check box.

- **Settings.** Use this area to set additional print variables. Define the number of copies to be plotted by entering or selecting a value in the **Number of Copies** edit box. Then, use the available check boxes to specify additional plot settings. Pick the **Rotate By 90 Degrees** check box to rotate the drawing plot 90 degrees on the sheet. Select the **All Colors As Black** check box if you want all drawing objects to be plotted in black, even if the drawing contains multiple colors. When you pick the **Remove Object Line Weights** check box, all drawing geometry is printed using the same line weight, even if the drawing contains objects with multiple line weights.

Figure 19-48.
The **Print Drawing** dialog box for printing a drawing.



- **Scale.** Use this area to specify the drawing print scale. Select the **Model 1:1** radio button to print the drawing sheet full-size, at scale. If the paper is too small for the specified sheet size, part of the sheet will not be displayed, but the views will be printed at full scale. When the **Model 1:1** radio button is selected, the **Tiling Enabled** check box becomes available, and if selected, sheets are titled, or arranged, for printing on multiple pages. If you do not want to apply a certain scale factor, select the **Best Fit** radio button, which scales the sheet according to the specified paper size. To apply a scale factor other than 1:1, pick the **Custom** radio button, and specify the scale in the **Custom** edit box. For example, if you enter a value of 2, the drawing is printed at a 2:1 scale. Or, if you enter a value of .5, the drawing is printed at a 1:2 scale. You can also choose the **Current Window** radio button to print the display of the current window without a specified scale factor.

Once you fully define the print options in the **Print** dialog box, select the **Preview...** button to display the **Print Preview** dialog box previously described and preview the intended print job. Finally, pick the **OK** button to print the drawing.

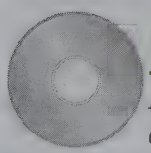
PROFESSIONAL TIP



Selecting the **Preview...** button in the **Print** dialog box is not required, though it is very good practice to preview your print job before it is sent to the printer, especially for large drawings that take a long time to print and use a large sheet of paper.



Exercise 19-9 Complete the Exercise on the Student CD.



Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. Discuss the contents of a complete set of working drawings.
2. Identify the *working drawing file type*, and explain why a single drawing file can be used.
3. Define *assembly drawing.*
4. Describe a *general assembly drawing.*
5. Explain a *detail assembly drawing* and give another name for this type of drawing.
6. What is an *erection assembly drawing*?
7. Discuss *subassembly drawings*, and briefly explain how they are created.
8. Describe what is shown in a *pictorial assembly.*
9. Give a complete definition of an *exploded assembly drawing.*
10. Give the steps recommended in this chapter for the creation of a set of working drawings.
11. What are *overlay drawing views* used to create and what is another name for overlay drawing views?
12. List the steps used for developing an overlay drawing view, which begins in the assembly model environment.
13. Briefly describe a *weldment drawing* and its relationship to the model.

14. Give general information about a *parts list.*
15. What are *balloons* and how are they used?
16. Do you add a parts list before or after you place balloons?
17. Describe *grouped balloons.*
18. List at least three items found in a typical parts list.
19. Identify the purpose of the green buttons displayed at each corner when you select the parts list.
20. Briefly discuss the automatic display of welding symbols and annotations, and their parametric relationship.
21. What is an *end fill*?
22. Define *caterpillar.*
23. Describe the purpose of the *reference line.*
24. Discuss and give an example of multiple sheets used in a set of working drawings, and explain the number of files involved.
25. Briefly discuss the fundamental difference between making a print of a model and a drawing.

Problems

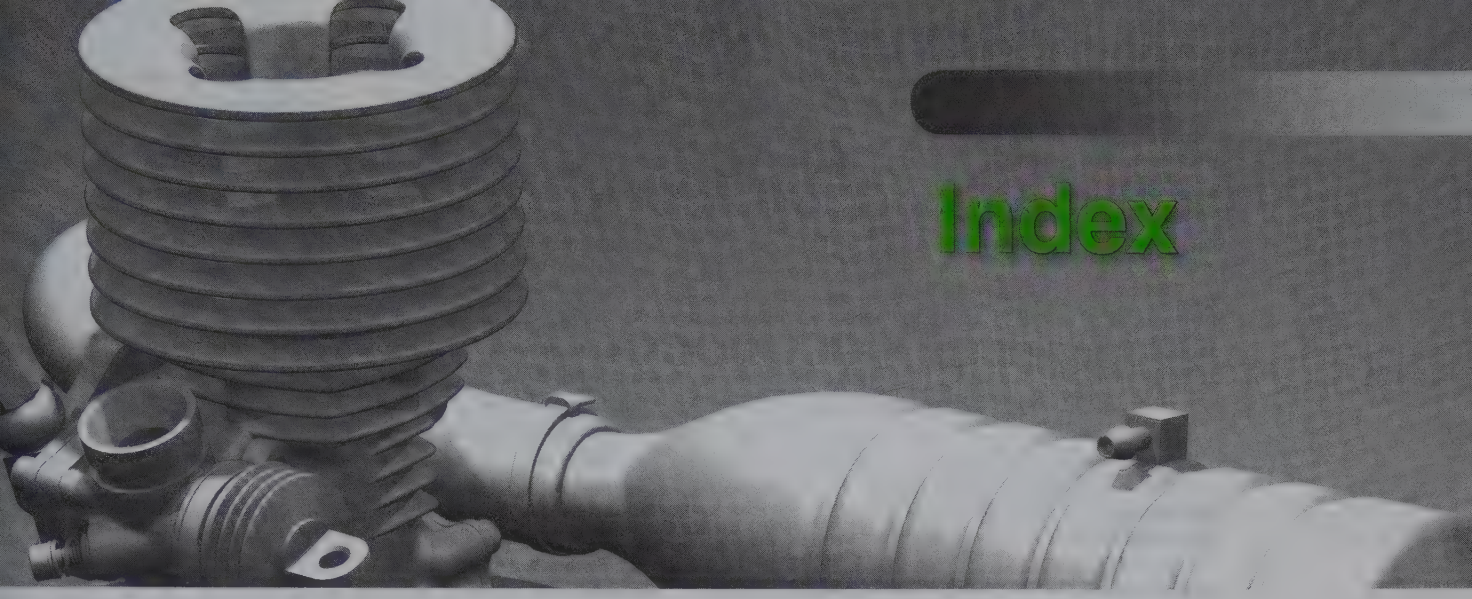
Instructions:

- Begin a new drawing file for each of the projects.
- Create a complete set of working drawings using each of the specified files.
- The first sheet should be an assembly with balloons and a parts list.
- Additional sheets are created for each of the details.
- Use a border and title block for each drawing sheet.
- Fully dimension, note, and annotate each drawing as required.
- Define the drawing properties in the drawing and individual model **Properties** dialog box, so the properties are displayed in the title block.

1. **Name:** Plate and Angle Drawings
   Units: Inch
   Template: DRAWING-in.idw
   Sheet Size: D
   Drawing View Scale: 1:1
   Assembly File: (Presentation) EX15-4.ipn
   Detail Files: EX14-4.ipt and EX14-5.ipt
   Special Instructions:
   - The assembly drawing is an exploded assembly drawing and is created by referencing a presentation file. Insert the presentation drawing view, using an isometric right display.

   Save As: P19-1.idw
2. **Name:** Hub Drawings
   Units: Inch
   Template: DRAWING-in.idw
   Sheet Size: B
   Drawing View Scale: 1:1
   Assembly File: EX12-17.iam
   Detail Files: EX7-3.ipt and EX12-17.ipt
   Save As: P19-2.idw

3. **Name:** Bottle Drawings
   Units: Inch
   Template: DRAWING-in.idw
   Sheet Size: B
   Drawing View Scale: 1:1
   Assembly File: P12-1.iam
   Detail Files: P6-8.ipt and P7-8.ipt
   Save As: P19-3.idw
4. **Name:** Rotating Hinge Drawings
   Units: Metric
   Template: DRAWING-mm.idw
   Sheet Size: A3
   Drawing View Scale: 1:1
   Assembly File: P12-3.iam
   Detail Files: P12-3a.ipt, P12-3b.ipt, and P12-3c.ipt
   Save As: P19-4.idw
5. **Name:** Housing Drawings
   Units: Inch
   Template: DRAWING-in.idw
   Sheet Size: C
   Drawing View Scale: 1:1
   Assembly File: P12-6.iam
   Detail Files: P11-2.ipt and P12-6.ipt
   Save As: P19-5.idw



Index

2D Chamfer dialog box, 116–117
2D Fillet dialog box, 115–116
2D Sketch Panel bar, 106
3D Intersection Curve dialog box, 328
3D Intersection tool, 328
3D intersections, sketching, 328
3D Line tool, 324
3D lines, sketching, 325–326
3D loft and sweep features, 323–324
3D Move/Rotate dialog box, 297–299
3D Sketch Bend dialog box, 326–327
3D sketch tools, 324–331
3D splines, sketching, 325–326

A

active standard, 635
adapting sketches in-place, 542–543
adaptive
    assembly parts, 548–551
    extrusions, 545
    features, 544–547
    holes, 546–547
    models, 539–553
    revolutions, 546
    sketches, 540–543
    subassemblies, 551–553
    work planes, 547
Adaptive icon, 539–540
adjusting components, 483–491
aligned dimensions, 730
aligned section, 687–688
alt-drag, 458–460
alternate position drawing views, 786
angle constraint, 447–448
angular dimensions, 731–732
Animate tool, 571, 573–575
animation, 570–575
    browser sequence views, 571–573
    interval, 573
    repetition, 573
Animation dialog box, 573–575
annotations
    orphaned, 639
    visibility, 678
application options, 89–95, 643–645
Apply Design View Representation dialog box, 784–785
arcs, sketching, 111–114
arrowless dimensioning, 743
arrow side, 514
aspect, 588, 601
assemblies, 18, 431–499
    adjusting components, 483–491
    alternative methods for inserting assembly components, 435–436
    constraining, 442–472
    converting to weldments, 530–531
    copying components, 480–483
    creating flexible subassemblies, 491–492
    creating parts in-place, 438–439
    creating subassemblies in-place, 440–442
    creating using existing components, 433–437
    developing assembly models, 432
    developing components in-place, 437–442
    inserting shared content, 436–437
    managing large, 498
    patterning components, 472–476
    positional representations, 494–497
    sectioning, 488–491
assembly constraints, 43, 543
    adapting assembly parts, 548–549
    adapting subassemblies, 551–553
    angle, 447–448
    insert, 451–453
    mate and flush, 444–447
    placing, 443–453
    tangent, 449–451
assembly drawings, 779–789
    dimensioning, 790–813
assembly drawing views
    creating, 782–784
    section views, 785
    working with, 784–786
assembly file, 17
assembly tools, 432
Auto Balloon dialog box, 794–795
Auto Balloon tool, 791, 794–795
Auto Dimension dialog box, 137
Autodesk Inventor Launchpad, 47
Automated Centerlines dialog box, 640–641, 724–725
automatic constraints, 128
automatic mapping, 315–316
automatic model edge referencing, 172–173
Autoproject edges during curve creation tool, 172, 307, 318

Autoproject edges for sketch creation and edit tool, 172, 307, 318
autoprojected feature edge, 172
auxiliary views, 682–685
Auxiliary View dialog box, 683
Auxiliary View tool, 682–683
axis of rotation, 125

B

Balloon All tool, 795
balloons, 591, 791
  adding, 791–795
  editing individual, 797
  modifying, 795–798
  styles, 591–594
Balloon tool, 791–793, 804
base feature, 22, 157
  sketching on, 171–173
base views, 670
  creating, 670–679
  custom views, 672–674
  labels, 674
  manipulating drawing view characteristics, 677–678
  measuring geometry, 677
  moving, 676
  projecting, 679–681
  scale, 674
  selecting multiple objects, 676–677
  working with, 676–679
baseline dimensioning, 733–735
Baseline Dimension Set tool, 733–734
Baseline Dimension tool, 733–734
Base View tool, 670
Bend dialog box, 417
bend relief, 384
bends, 416–417
Bend tool, 326
BOM Properties dialog box, 792–793, 800
Border dialog box, 653
borders
  custom, 649–653
  default, 646–649
boundary, 700–703
Break Out View dialog box, 694–697
Break Out View tool, 693–697
break, 703–707. *See also* conventional breaks
Broken Out View tool, 694
Broken View dialog box, 704–705, 707
Broken View tool, 704
broken-out section views, 693–699
  creating, 694–698
  From Point option, 694
  modifying, 698–699
  Through Part option, 697
  To Hole option, 696–697
  To Sketch option, 695–696
broken-out sections. *See* broken-out section views
Browse for Folder dialog box, 61
browser, 34–35
browser bar, 34–35

C

camera view, 569
cascading submenu, 28
catalog features, 22, 24, 359–377. *See also* iFeatures
caterpillars, 809–811
Caterpillar tool, 809
Centered Pattern tool, 727
Centerline Bisector tool, 727–728
center lines, 311
centerlines, 594
  adding, 724–727
Centerline tool, 725–726
center marks, 594
  adding, 724–727
  styles, 594–595
Center Mark tool, 725–726
Center point arc tool, 112, 113
Chamfer dialog box, 222–226
  Distance option, 222
  More area, 223–224
Chamfer Note tool, 737, 739–740
chamfers, 116–117, 222
  creating, 222–226
  dimensioning, 739–740
Chamfer tool, 116–117, 222–223, 512, 530
child nodes, 34
Choose project file dialog box, 59
circles, sketching, 109–111
circular feature pattern, 263–267
circular pattern, 25
Circular Pattern dialog box, 263–267
Circular Pattern tool, 125, 263–264, 512, 530
circumscribed polygon, 119
class, 189
Coil dialog box, 164–169
  Coil Ends tab, 168–169
  Coil Shape tab, 164–166
  Coil Size tab, 167–168
coils
  creating additional, 179
  developing, 164–170
Coil tool, 164–169, 190, 234
colors, changing, 486–488
color styles, 80–83
Column Properties dialog box, 769–770
combs, 107
Common View tool, 73
complex features, 307–346
component, 18
component pattern, 472
components
  adjusting, 483–491
  changing colors, 486–488
  copying, 480–483
  mirroring, 477–480
  moving, 485
  patterning, 472–476
  replacing, 483–484
  rotating, 485–486
composite iMate, 462–463
Connect to Printer dialog box, 818
constant fillet, 213–214
constant round, 213–214
constraining sketches using dimensions, 131–140
constraints, 25
  assembly, 442–472
  automatic, 128
  constraining components in-place, 460
  dragging components, 458–460
  driving, 469–472
  editing, 468
  geometric, 128–130
  mate, 444–447
  placing, 443–458

context sensitive, 31
contour flange, 392–401
Contour Flange dialog box, 393–397
Bend area, 397
Extents area, 394–397
Shape tab, 393–394
control keys, 37
conventional breaks
creating, 703–707
editing, 707
Convert to Weldment dialog box, 530–531
Copy Components dialog box, 481
Copy Components tool, 481–482
Copy Components: File Names dialog box, 482
copying components, 480–483
corner chamfer, 419–421
Corner Chamfer dialog box, 419–421
Corner Chamfer tool, 419
corner relief, 386
corner rip, 413, 415
corner round, 417–419
Corner Round dialog box, 418–419
Corner Round tool, 417–418
corner seam, 411–415
Corner Seam dialog box, 411–415
Corner Options tab, 415
Shape tab, 411–415
Corner Seam tool, 411, 415
cosmetic weld, 516, 521–523
Cosmetic Weld dialog box, 521–523
Cosmetic Weld tool, 521, 523, 805
counterbored hole, 182
countersunk hole, 182
Create Component tool, 437–438, 440, 542
Create iFeature dialog box, 361–366
Position Geometry area, 365
Selected Features area, 363, 365
Size Parameters area, 363–365
Create iMate dialog box, 461–462
Create iMate tool, 461
Create In-Place Component dialog box, 440–441
Create In-Place dialog box, 438
Create View tool, 559, 575
cursor menu, 31
custom borders
adding text, 649, 651
filling and hatching sketch regions, 652–653
custom drawing symbols, 764–766
custom templates, 55–56
Custom View window, 672–674
Custom/Virtual Parts dialog box, 804
Cut dialog box, 398–401
cut edges, projecting, 289
cut extrusion, 174, 175
cutting-plane lines, 686–692
modifying, 692
Cut tool, 398

D

dangling geometry, 423
datum, 734, 757
datum dimensioning, 733–735
datum feature symbol, 621–622
placing, 757–759
datum identification symbol, 621
placing, 757–759
Datum Identifier Symbol tool, 758–759
Datum Target – Circle tool, 759–761
Datum Target – Leader tool, 759–760
Datum Target – Line tool, 759–762
Datum Target – Point tool, 759–760, 762
Datum Target – Rectangle tool, 759–760, 763
datum targets, 595, 759
datum target styles, 595–598
datum target symbols, 759–763
decal, 341–346
Decal dialog box, 345
decal image sketch, 342, 344–345
Decal tool, 341–342, 345–346
Default Drawing Border Parameters dialog box, 646–649
dependent views, deleting, 681
depth, 514
derived component, 24
design elements, 24, 359. *See also* iFeatures
design view representation, 492–494
Design View Representations dialog box, 493–494
detail assembly drawing, 780
detail view, 699–703
Detail View dialog box, 699–701
Detail View tool, 699
dialog boxes, 28, 30–31
diameter dimensions, 731
Dimension Properties dialog box, 138–140
Dimension Settings tab, 138–140
Document Settings tab, 140
dimensions, 131, 721
constraining sketches using, 131–140
dimensioning
arrowless, 743–744
assembly and weldment drawings, 790–813
baseline, 733–735
centerlines and center marks, 724–727
Chamfer Note tool, 739–740
custom drawing symbols, 764–766
datum, 733–735
drawing dimensions, 724–772
General Dimension tool, 728–733
geometric dimensioning and tolerancing, 754–763
hole and thread notes, 738
model dimensions, 722–723
modifying dimensions, 771–772
ordinate, 743–744
part drawings, 721–772
revision table and revision tags, 767–771
surface texture symbols, 741–742
tables, 751–754
tabular, 745–751
text, 736–737
dimension styles, 598–610
docking, 33, 35
document settings, 89, 638–642
Document Settings dialog box, 89
Drawing tab, 638–642
Modeling tab, 189–190, 324, 551
Sheet tab, 642–643
Sketch tab, 324, 638–639
Standard tab, 79–80, 638
Draft View dialog box, 708
dragging, 143–144
drag-mate, 458–460
drawing annotation tools, 721
drawing dimensions, 724–772
drawing files, 19
drawings, developing, 580–581
drawing sheet, 581
parameters, 645–646
Drawing Sketch Panel toolbar, 649–650

drawing styles, 581–582
Drawing View dialog box, 670–671, 782–784
Component tab, 670–671, 782–783
Display Options tab, 671–672, 784
Model State tab, 671, 783, 784, 790
drawing views, 669–710
drawing view tools, 669
drilled hole, 182
Drive Constraint dialog box, 469–472
driven constraint, 469–472
driven dimensions, 135–136

E

edge fillets, 211–217
edge rounds, 211–217
Edit Constraint dialog box, 468–469
Edit Dimension dialog box, 131–133, 468, 723–724
Edit Hole Table dialog box, 751
Edit iFeature dialog box, 369
Edit Parts List dialog box, 801–805
Edit Sheet dialog box, 645–646
Edit Table dialog box, 751–754
Edit Task & Sequences dialog box, 572
ellipses, sketching, 109–111
embossing, 335–339
Emboss dialog box, 336–338
Emboss tool, 335–336
End Fill dialog box, 807–808
end fills, 528–529, 807
placing, 807–809
End Fill tool, 528–529
End of Part, 34–35
engraving, 335–339
erection assembly drawing, 780
expert mode, 33
exploded assembly, 557
exploded assembly drawing, 781
exploding, 561–562
exploding presentations, 561–568
Export Parts List dialog box, 802–803
Extend tool, 144
external threads, 233
Extract iFeature tool, 361
extruded base feature, 157–162
Extrude dialog box, 157–161
creating additional extrusions, 173–179
More tab, 161
open sketch profiles, 180
Shape tab, 157–160
Extrude tool, 157, 173, 512, 530
extrusions
adapting, 545
creating additional, 173–179

F

Face dialog box, 388–391
Relief Options tab, 390–391
Shape tab, 388–390
Unfold Options tab, 390–391
face draft, applying, 230–233
Face Draft dialog box, 230–233
face fillet, 218
face round, 218
faces, moving, 239–241
Face tool, 388
Feature Control Frame dialog box, 755–756
Feature Control Frame tool, 755–756
feature control frame, 611
adding, 754–756
style, 611–614
feature identifier symbols, 757
feature patterns, 25, 253–267
circular, 263–267
rectangular, 257–263
suppressing, 262, 267
Feature Properties dialog box, 545–547
fence, 700–701
Field Entry dialog box, 651
File Open Options dialog box, 44–45, 560
files, 17–20
finding, 45–46
opening existing, 44–46
starting new, 37–38
fill direction, 524–526
Fillet dialog box, 210–221
Constant tab, 210, 213–214
Face tab, 218
Setbacks tab, 216–217
Variable tab, 215–216
fillets, 115–116, 210
adding, 210–221
constant, 213–214
edge, 211–217
variable, 213, 215–217
Fillet tool, 115–116, 209–211, 214, 512, 530
fillet weld, 516–520
Fillet Weld dialog box, 516–520
Fillet Weld tool, 516, 805
filling sketch regions, 652–653
Find: Inventor Files dialog box, 45–46
flange, 404
adding, 404–407
Flange dialog box, 405–407
Extents area, 406–407
Shape tab, 405–406
Flange tool, 404–405, 407
flat end, 168
Flat Pattern Extents dialog box, 425
flat patterns, 424–426
Flat Pattern tool, 424–426
flexible subassembly, 491–492
floating window, 33
flush constraint, 444–447
flyout, 36
Fold dialog box, 401–403
folds, 401–403
Format Column dialog box, 619, 625–627
Format Text dialog box, 120–122, 649–651, 736, 737, 758
full radius fillets and rounds, 219–221
full section, 686
fully-constrained model, 25

G

general assembly drawing, 780
General Dimension tool, 131–137, 728–733, 737
general notes, 736
geometric constraints, 128–130
geometric dimensioning and tolerancing (GD&T), 754–763
datum feature symbols, 757–759
datum target symbols, 759–763
feature control frames, 754–756
feature identifier symbols, 757
revision table and revision tags, 767–771

geometry, referencing and projecting, 102, 326, 328
grab bar, 33
graphical user interface (GUI), 26
grips, 107
groove weld, 516, 523–527
    fill direction, 524–526
Groove Weld dialog box, 523–527
Groove Weld tool, 516, 523, 805
Grounded Work Point tool, 297–299
grouped balloons, 795
guide rail, 321
guide surface, 321

H

half section, 489, 687
Hatch/Color Fill dialog box, 652–653
hatching, 615
    sketch regions, 652–653
hatch styles, 615–616
height, 168
help, 47–48
help string, 27–28, 36
hem, 408
Hem dialog box, 408–410
hierarchical menu, 28
hole and thread notes, 738–739
hole centers, 118–119
holes, 182
    adapting, 546–547
    creating, 182–191
Holes dialog box, 182–191, 233
    Concentric placement option, 184–185
    From Sketch placement option, 182–183
    On Point placement option, 185–186
hole table
    modifying, 749–751
    rows and columns, 749
    styles, 617–620
Hole Table Column Chooser dialog box, 618
Hole Table – Selected Feature tool, 745, 747–748
Hole Table – Selection tool, 745–747
Hole Table – View tool, 745, 747–748
hole tags, 748–749
Hole/Thread Notes tool, 737, 738
Hole tool, 182, 512, 530
hot keys, 37

I

.iam file, 18, 38, 559
.ide file, 369
identification line, 520, 811–812
i-drop, 437
ID styles, 621–622
.idw file, 19, 580
iFeature Author dialog box, 369–375
    Geometry tab, 371
    Other tab, 373
    Parameters tab, 370–371
    Properties tab, 371–372
    Threads tab, 372–373
iFeature Author Table tool, 369
iFeature factories, 375–376
iFeature factory table, 373–375
iFeature file, 369
iFeatures, 24, 359. *See also* catalog features
    creating, 361–366
    editing, 368–369
    inserting, 366–368
    table-driven, 369–376
    viewing and managing, 360–361
illustrated parts breakdown, 562
Image Properties dialog box, 344
iMates, 191, 461
    creating, 461–466
    using, 466–468
Include Geometry tool, 326, 328
Incremental View Rotate dialog box, 570, 674
Infer iMates dialog box, 465
in-place, 437
inscribed polygon, 119
insert constraint, 451–453
Insert iFeature dialog box, 366–368, 376
Insert Image tool, 341, 342, 344
intelligence, 21
interface, 26–37
internal threads, 233
intersection extrusion, 174, 175
Inventor Precise Input toolbar, 140–141, 325
Inventor project wizard dialog box, 64–65
Inventor Standard toolbar, 35–36
    Color drop-down list, 79–80
.ipj file, 57
.ipn file, 19, 559
iProperties, 67–71, 642
.ipt file, 17, 38, 436, 437
Isometric tool, 76

K

keys, 370–371

L

large assemblies, managing, 498
layers, 586–587
leader, 587–588, 737
leader notes, 737–738
leader styles, 587–589
Leader Text tool, 737–738
learning mode, 32
leg, 514
lighting styles, 84–85
Line tool, 103–104, 325, 326
    creating arcs, 114
linear dimensions, 728, 730
lines, sketching, 103–105
List Values dialog box, 364–365
local notes, 736
Loft dialog box, 308–316
    Conditions tab, 311–315
    Curves tab, 308–309
    Transition tab, 315
lofts, 307
    advanced, 323–324
    creating, 307–317
Loft tool, 308, 324
Look At tool, 76

M

map points, 315–316
mate constraint, 444–447
material styles, 85–86
maximized, 39
Measure tool, 160
menu access keys, 28

Mirror Components dialog box, 477, 483
Mirror Components: File Names dialog box, 479–480
Mirror Components tool, 477–479
Mirror dialog box, 123
mirrored component, 477
mirrored feature, 25, 254–257
mirroring
components, 477–480
sketch objects, 123
Mirror Pattern dialog box, 254–255
mirror plane, 254, 477
Mirror symbol, 478–479
model dimensions, 722–723
modeling failure, 25
model parameters, 122
Modify Hatch Pattern dialog box, 691–692, 699
monodetail drawing, 669
motion constraints, 443
adding, 453–457
Move dialog box, 145
Move Face dialog box, 239–240
Move Face tool, 239–240, 512, 530
Move tool, 145–146
moving components, 485
multidetail drawing, 669
multiple design interface, 40
multiple document interface, 40
multiple position drawing views. *See* alternate position drawing views
multiple sheet drawings, 814–816

N

natural end, 168
New Sheet dialog box, 814
New Style Name dialog box, 583
nominal, 138

O

object defaults, 634–635
occurrences, 257
offset, 234
Offset dialog box, 282
offset section, 687
Offset tool, 127
offsetting, 236–239
sketch geometry, 127
Open dialog box, 433–434
New File window, 37–38
Open File window, 37–38, 45
Projects icon, 57
Projects window, 64
opening existing files, 44–46
open sketch profile, using, 180–181
Options dialog box, 89–95
Assembly tab, 543, 547
Colors tab, 91–92
Display tab, 92–93
Drawing tab, 643–645, 722–723
General tab, 90–91
iFeature tab, 360–361, 366
Part tab, 94–95, 340, 438–439, 440
Sketch tab, 93–94, 324
ordinate dimensioning, 743–744
Ordinate Dimension Set tool, 743–744
orphaned annotations, 639
other side, 514
over-constrained model, 25
overlay drawing views, 786–789
Overlay View dialog box, 788–789
Override Object dialog box, 495–497

P

panel bar, 31–33
Pan tool, 74
parameters, 141
Parameters dialog box, 141–143, 363, 393
parametric solid modeling, 21–22
parent node, 34
part drawing views, 669–710
part file, 17
part model, 22–25
part model view, 669
partial auxiliary views, 683, 685
parts list, 591, 622, 791
exporting, 802–803
inserting, 799–800
modifying, 801–805
spreadsheet, 803–805
styles, 622–627
Parts List Column Chooser dialog box, 624
Parts List dialog box, 800
Pattern Component dialog box, 473–476
Associative tab, 473–474
Circular tab, 475–476
Rectangular tab, 474–475
Pattern Component tool, 472
patterning sketch geometry, 122–126
perspective drawing view, 673
pictorial assembly drawing, 781
pitch, 167, 189, 514
Place Component tool, 433, 440
iMates, 466–468
Place Constraint dialog box, 443–453
Assembly tab, 443–453
Motion tab, 453–457
Transitional tab, 457–458
Place Constraint tool, 443–453, 455, 457
iMates, 467–468
placed features, 22, 209–241
sheet metal, 404–421
Point tool, 118
polygon, 119
Polygon tool, 119–120
pop-up menus, 31
positional representations, 494–497
precise input, 140–141
Precise View Rotation tool, 569–570
Precision and Tolerance dialog box, 608–610
prefix, 520, 602, 813
Preparations group, 511, 514
presentation file, 19
Presentation Panel bar, 558
presentations, 557–576
animating, 570–575
creating views, 559–561
exploding, 561–568
placing additional views, 575–576
Print Drawing dialog box, 818–819
printing drawings, 817–820
Print Preview dialog box, 818–819
Print Setup dialog box, 817–818
Project Curve to Surface dialog box, 329
Project Cut Edges tool, 289
Project Editor, 57–65
Projected View tool, 679–680
Project folder, 60–63
Project Geometry tool, 289

projecting
curve onto surface, 329–332
cut edges, 289
views, 679–681
projects, 56–65
creating new, 64–65
Properties dialog box, 67–72, 642, 799
Custom tab, 71, 655–656
General tab, 68
Occurrence tab, 548, 552
Physical tab, 71–72, 532
Project tab, 69, 70
Save tab, 71, 72
Status tab, 69–70
Summary tab, 68–69
Weld Bead tab, 532–533
prototype, 37
pull direction, 231
pull-down menus, 27–28
PunchTool dialog box, 421–423
Purge Styles dialog box, 88–89

Q

Quantity Note dialog box, 609
quarter section, 489

R

radial dimensions, 730
rail, 310–311, 321
rectangles, sketching, 114–115
rectangular feature pattern, 257–263
rectangular pattern, 25
Rectangular Pattern dialog box, 124, 257–263
Rectangular Pattern tool, 124, 258, 512, 530
reference line, 520, 811
reference sketch, 542–543
referencing and projecting geometry, 102, 326, 328
referencing existing geometry, 102, 104–105
regular polygons, sketching, 119–120
Replace All tool, 483–484
Replace Component tool, 483–484
replacing components, 485
Resolve Link dialog box, 434
Retrieve Dimensions dialog box, 723
revision block. *See* revision table
revision history block. *See* revision table
revision symbol. *See* revision tag
revision table, 767–770
Revision Table tool, 767–768
revision tag, 767, 770–771
Revision Tag tool, 770–771
revolutions, 162, 167
adapting, 546
creating additional, 179
revolved base feature, creating, 162–164
Revolve dialog box, 162–164, 180
Revolve tool, 162–164, 512, 530
Rib dialog box, 191–195
rib or web network, 191
ribs, 191–195
rigid body, 548
root opening, 511
Rotate Component tool, 486
Rotate dialog box, 146
Rotate tool, 73, 146
Rotate View dialog box, 677–678
rotating components, 485–486
rounds, 115–116, 210
adding, 210–221
constant, 213–214
edge, 211–217
variable, 213, 215–217

S

SAT files, 434, 437
Save As dialog box, 43–44
Save Copy As dialog box, 42–43
Save Styles To Styles Library dialog box, 87–88
Save tool, 42
saving work, 42–43
search tool, 45–46
section lines, 691–692
Section View dialog box, 688–690
Section View tool, 685–686, 689, 691
section views, 685–692
sectional views. *See* section views
sectioning assemblies, 488–491
Select Assembly dialog box, 559–560
sequence, 571
Sequence View filter option, 571–573
setbacks, 215–217
shared content, 436–437
sheet, 581
creating new blank, 814
reusing existing, 815
parameters, 645–646
sheet formats, 709–710, 815
sheet labels, 642
sheet metal face, 388–391
sheet metal parts, 381–426
bends, 416–417
contour flanges, 392–401
corner chamfers, 419–421
corner rounds, 417–419
corner seams, 411–415
creating, 381–382
cuts, 398–401
faces, 388–391
flat patterns, 424–426
folds, 401–403
hems, 408–410
placed features, 404–421
PunchTool tool, 421–423
setup and format, 382–388
sketched features, 388–403
sheet metal styles, 382–388
Sheet Metal Styles dialog box, 382–388, 390–394, 399, 403, 408–409, 411, 415
Bend tab, 384–386
Corner tab, 386–387
Sheet tab, 382–384
shell, 226
Shell dialog box, 226–230
More tab, 228–230
shelling features, 226–230
shortcut keys, 37
shortcut menu, 31
Show Constraints tool, 130
Show Dimensions tool, 160
size, 514, 588, 601
Sketch Doctor dialog box, 147–148
sketches, 22, 99
beginning, 101
concepts, 99–148
constraining using dimensions, 131–140

sketches *(continued)*
constraining using geometric constraints, 128–130
finishing, 101
sketched features, 22, 157–195
additional extrusions, 173–179
additional revolutions and coils, 179
coil, 164–170
extruded base features, 157–162
holes, 182–191
open sketch profiles, 180–181
revolved base features, 162–164
ribs and webs, 191–195
sketching on a base features, 171–173
sketch editing tools, 143–147
Sketched Symbol dialog box, 764–765
sketching
3D intersections, 328
3D lines and splines, 325–326
adding fillets, rounds, and chamfers, 115–117
adding text, 120–122
arcs, 111–114
circles and ellipses, 109–111
concepts, 99–101
constraining sketches, 128–140
lines, 103–105
on a base feature, 171–173
projecting geometry, 102
rectangles, 114–115
referencing existing geometry, 102, 104–105
regular polygons, 119–120
sketch dimension properties, 138–140
sketch editing tools, 143–147
sketch parameters, 141–143
splines, 106–109
views, 708
sketch patterns, 122–126
circular, 125–126
mirrored, 123
rectangular, 124–125
sketch points, 118–119
solid model, 21
Sort Parts List dialog box, 802–803
Sort Table dialog box, 753, 754
spacing, 259
specific notes, 736
Specify Range dialog box, 364, 375
splines, 106
3D, 325–326
bowtie options, 107, 108
editing, 107–109
sketching, 106–109
Spline Tension dialog box, 109
Spline tool, 106–107, 325, 332
split, 332
Split dialog box, 333–335
Split Face tool, 333
split features, 332–335
Split tool, 332–334
spreadsheet, 375–376
parts list, 803–805
standards, 635–637
status bar, 27, 29
style library, 87
styles
activating, 79
purging, 88–89
updating, 87
styles and standards, 77–89, 581–637
controlling, 637
Styles and Standards Editor, 78–89, 582–586
Balloon Formatting area, 592–594
Center Mark Style area, 594–595
Datum Target Style area, 595–598
Dimension Style area, 598–610
Alternate Units tab, 599–600
Display tab, 600–601
Notes and Leaders tab, 606–610
Options tab, 605–606
Text tab, 601–603
Tolerance tab, 603–605
Units tab, 598–599
Feature Control Frame Style area, 611–614
Hatch Style area, 615–616
Hole Table Style area, 617–620
ID Style area, 621–622
Layer Styles area, 586–587, 634
Leader Style area, 588–589
Object Default Style area, 632–633
Parts List Style area, 622–627
Standard area, 635–637
Surface Texture Style area, 627–628
Table Style area, 629–630
Text Styles area, 590–591
Weld Bead Style area, 632–633
Weld Symbol Style area, 631–632
subassemblies, 18, 431, 440
adapting, 551–553
creating in-place, 440–442
flexible, 491–492
subassembly assembly drawing, 780–781
suffix, 602–603
surface, 159, 339
opaque, 340
surface finish, 627, 741–742
Surface Texture dialog box, 741–742
surface texture styles, 627–628
Surface Texture Symbol tool, 741
surface tools and techniques, 339–341
Sweep dialog box, 318–321
sweeps, 317
advanced, 323–324
creating, 317–322
Sweep tool, 318, 324, 512, 530
symbols, custom, 764–766
Symbols dialog box, 765–766

T

Table Column Chooser dialog box, 753
Table dialog box, 751–752
table-driven iFeatures
creating, 369–375
inserting, 376–377
tables, adding, 751–754
table styles, 628–630
Table tool, 628–629, 751–752
tabular dimensioning, 617, 745–751
Tangent arc tool, 112–113
tangent constraint, 449–451
templates, 37, 53–56
creating, 54–55
Templates folder, 54
text
adding to sketch, 120–122
adding to drawing, 736–737
text leader, 737
text styles, 589–591
Text tool, 649, 651, 736–737
Texture Chooser dialog box, 82

thickening, 236–239
Thicken/Offset dialog box, 236
More tab, 238–239
Thicken/Offset tab, 236–238
third-party content, 436–437
thread class, 189
Thread Feature dialog box, 233–235
Location tab, 234
Specification tab, 234–235
Thread spreadsheet, 235
Thread tool, 170, 233–234
threads, placing, 233–235
Three point arc tool, 112
Three point rectangle tool, 114–115
three-quarter section, 489
Title Block dialog box, 656–657
title blocks, 653
custom, 657–658
default, 654–656
tolerance, 603
toolbar buttons, 35
toolbars, 35–36
tool groups, 35
tooltip, 36
trails, 19, 560
editing, 566–568
transitional constraints, 443
adding, 457–458
tree structure, 34
Trim tool, 144
Tweak Component dialog box, 562–565, 568
tweaking, 19, 562–566
tweaks, 562
editing, 566–568
Two point rectangle tool, 114

U

under-constrained model, 25
Unfold Method List dialog box, 383
Update Styles dialog box, 87
user-interface, 26–37
user parameters, 122

V

variable fillet, 213
forming, 215–217
variable round, 213
forming, 215–217
View Catalog dialog box, 360
view options, 72–77
View pull-down menu, 73–77
virtual components, 438
Visual Syllabus dialog box, 48

W

webs, 191–195
Weld Bead Report tool, 533–534
weld bead styles, 632–633
Weld Caterpillars dialog box, 809–811
weld length, 514
weld placement, 514–529. *See also* welding
Welding Symbol dialog box, 527–528, 811–813
weld symbols
placing, 811–813
styles, 630–632
welding, 509, 514–529
cosmetic weld, 516, 521–523
fillet weld, 516–520
groove weld, 516, 523–527
terminology, 514
welding symbols, 518–519, 522
adding, 527
weldment drawings, 630–631, 789–790
dimensioning, 790–813
weldment drawing views, 789–790
Weldment Features Panel bar, 512–513
weldment machining, 529–530
weldment model symbols and annotations, 805–806
weldments, 37, 509–534
converting assemblies to, 530–531
creating weld bead reports, 533–534
developing models, 510
editing, 532–533
fillet weld, 516–520
material preparation, 511–514
Welds group, 516
Window Control pull-down menu, 39
Window pull-down menu, 40–41
work axes, 22, 279
using, 290–293
Work Axis tool, 291–292
work axis, 290
work features, 22, 279–299
Workgroup Search Paths, 61–62
working drawings, 779, 780, 781
work planes, 22
adapting, 547
using, 279–289
Work Plane tool, 281–288, 477
work points, 22, 279, 294
using, 294–299
Work Point tool, 294–296, 299
Workspace, 61

Z

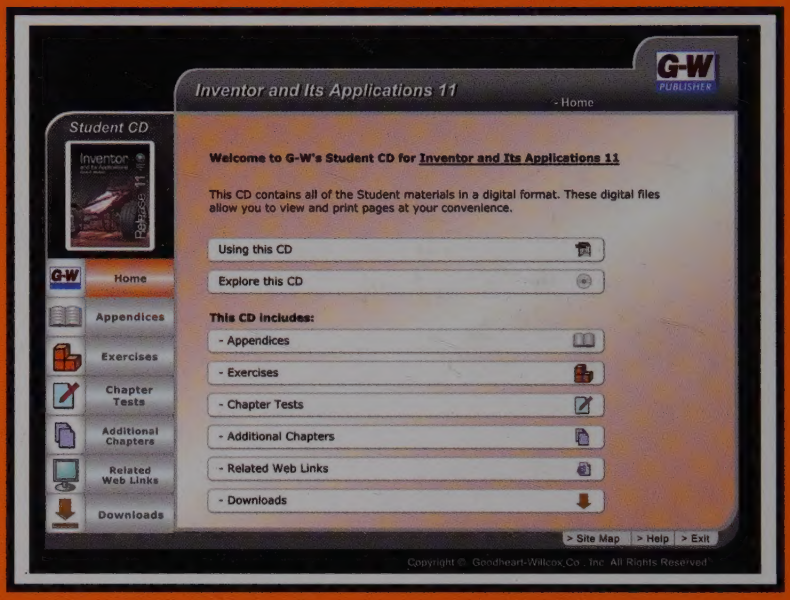
Zoom Select tool, 75

Inventor and Its Applications—Release 11

Inventor and Its Applications describes in detail the steps for creating sketches, models, and drawings using Autodesk Inventor®11. The text teaches students how to develop and control sketches with constraints, dimensions, and tables. Once these skills are mastered, the students use the sketches to build models of various complexities, then add features, create assemblies, and develop drawings. The text gives students an opportunity to show off their designs in the form of orthographic drawings, static and dynamic presentations, and photorealistic renderings. Inventor and Its Applications contains over 800 illustrations to reinforce the descriptions and methods covered in the chapters. This text not only teaches how to use Autodesk Inventor, but also provides skills that will help learners excel in a career in drafting, design, or engineering.

Features...

- ✔ Includes a Student CD containing Exercises, Supplemental Material, Related Web sites, Chapter Tests, and more.
- ✔ Heavily illustrated to make learning easy.
- ✔ Step-by-step use of Inventor tools.
- ✔ Exercises throughout the chapters reinforce Inventor concepts. End-of-chapter questions and problems check comprehension.
- ✔ Professional Tips explain how to use Inventor effectively and efficiently.



Other Goodheart-Willcox titles...

- **AutoCAD and Its Applications—Basics,** by Terence M. Shumaker, David A. Madsen, and David P. Madsen
- **AutoCAD and Its Applications—Advanced,** by Terence M. Shumaker, David A. Madsen, and Craig P. Black
- **AutoCAD and Its Applications—Comprehensive,** by Terence M. Shumaker, David A. Madsen, David P. Madsen, and Craig P. Black
- **Parametric Modeling Using Inventor,** by James P. LoFaso
- **AutoCAD LT Fundamentals,** by Ted Saufley and Paul B. Schreiner
- **Learning Autodesk VIZ: A Tutorial Approach,** by Sham Tickoo and David McLees
- **Learning Inventor: A Process-Based Approach,** by Thomas Short, Anthony Dudek, and Bill Kramer
- **VBA for AutoCAD,** by Paul Richard

Autodesk®
Authorized Publisher

AUTODESK INVENTOR® 11
Certified
Autodesk

ISBN 978-1-59070-775-3
90000
9 781590 707753

For more information about Goodheart-Willcox titles, visit our Web site at www.g-w.com.